FloEFD™ for Creo
Tutorial

Software Version 13

Rev. 09242013

© 2013 Mentor Graphics Corporation
All rights reserved.

This document contains information that is proprietary to Mentor Graphics Corporation. The original recipient of this document may duplicate this document in whole or in part for internal business purposes only, provided that this entire notice appears in all copies. In duplicating any part of this document, the recipient agrees to make every reasonable effort to prevent the unauthorized use and distribution of the proprietary information.

This document is for information and instruction purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in written agreements between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

RESTRICTED RIGHTS LEGEND 03/97

U.S. Government Restricted Rights. The SOFTWARE and documentation have been developed entirely at private expense and are commercial computer software provided with restricted rights. Use, duplication or disclosure by the U.S. Government or a U.S. Government subcontractor is subject to the restrictions set forth in the license agreement provided with the software pursuant to DFARS 227.7202-3(a) or as set forth in subparagraph (c)(1) and (2) of the Commercial Computer Software - Restricted Rights clause at FAR 52.227-19, as applicable.

Contractor/manufacturer is:
Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com
SupportNet: supportnet.mentor.com/
Send Feedback on Documentation: supportnet.mentor.com/doc_feedback_form

TRADEMARKS: The trademarks, logos and service marks ("Marks") used herein are the property of Mentor Graphics Corporation or other third parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the respective third-party owner. The use herein of a third-party Mark is not an attempt to indicate Mentor Graphics as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A current list of Mentor Graphics' trademarks may be viewed at: www.mentor.com/trademarks.

Contents

Features List FL-1

First Steps

Ball Valve Design

Opening the Model A1-1
Creating a FloEFD Project A1-2
Specifying Boundary Conditions A1-5
Specifying the Engineering Goal A1-7
Running the Calculation A1-8
Monitoring the Solver A1-9
Adjusting Model Transparency A1-10
Viewing Cut Plots A1-11
Viewing Surface Plots A1-14
Viewing Isosurface Plots A1-15
Viewing Flow Trajectories A1-16
Viewing XY Plots A1-17
Viewing Surface Parameters A1-18
Analyzing a Design Variant in the Creo Elements/Pro & Creo Parametric Ball

part. A1-18
Cloning the Project A1-19
Creating a New Instance in the Family Table A1-19
Changing the Model Geometry A1-20
Replacing the Instance in the Assembly A1-20
Analyzing a Design Variant in the FloEFD Application A1-22

Conjugate Heat Transfer

Opening the Model A2-1
Preparing the Model A2-2
Creating a FloEFD Project A2-3
Specifying the Fan A2-7
Specifying Boundary Conditions. A2-8
Specifying Heat Sources A2-9
Creating Solid Materials in the Engineering Database A2-11
Specifying Solid Materials A2-13
Specifying Engineering Goals A2-14
Changing the Geometry Resolution A2-18
Running the Calculation A2-19
Viewing the Goals A2-19
Adjusting Model Transparency A2-20
Viewing Flow Trajectories A2-20
Viewing Cut Plots A2-22
Viewing Surface Plots A2-24

Porous Media

Opening the Model A3-2
Creating a FloEFD Project A3-2
Specifying Boundary Conditions A3-4
Creating Isotropic Porous Medium in the Engineering Database A3-5
Specifying Porous Medium A3-7
Specifying Surface Goals A3-7
Specifying the Equation Goal A3-8
Running the Calculation A3-9
Viewing the Goals A3-10
Viewing Flow Trajectories A3-11
Cloning the Project A3-12
Creating Unidirectional Porous Medium in the Engineering Database . A3-12
Specifying the Porous Medium - Unidirectional Type A3-13
Comparing the Isotropic and Unidirectional Catalysts A3-13

Intermediate Examples

Determination of Hydraulic Loss

Opening the Model B1-1
Model Description B1-2
Creating a FloEFD Project B1-3
Specifying Boundary Conditions B1-7
Specifying Surface Goals B1-8
Running the Calculation B1-9
Monitoring the Solver B1-10
Cloning the Project B1-11
Viewing Cut Plots B1-11
Working with Parameter List B1-14
Viewing the Goal Plot B1-15
Working with Calculator B1-16
Changing the Geometry Resolution B1-18

Cylinder Drag Coefficient

Problem Statement B2-1
Opening the Model B2-2
Creating a FloEFD Project B2-2
Specifying 2D simulation B2-6
Specifying a Global Goal B2-7
Specifying an Equation Goal B2-7
Cloning the Project B2-8
Changing Project Settings B2-9
Changing the Equation Goal B2-9
Creating a Template B2-10
Creating a Project from the Template B2-11
Solving a Set of Projects B2-12
Getting Results B2-12

Heat Exchanger Efficiency

Problem Statement B3-1
Opening the Model B3-2
Creating a FloEFD Project B3-3
Specifying Symmetry Condition B3-5
Specifying a Fluid Subdomain B3-6
Specifying Boundary Conditions B3-8
Specifying Solid Materials B3-12
Specifying a Volume Goal B3-12
Running the Calculation B3-13
Viewing the Goals B3-13
Viewing Cut Plots B3-14
Adjusting the Parameter Display Range B3-15
Displaying Flow Trajectories B3-16
Viewing the Surface Parameters B3-18
Calculating the Heat Exchanger Efficiency B3-19

Mesh Optimization

Problem Statement B4-2
Opening the Model B4-3
Creating a FloEFD Project B4-3
Specifying Boundary Conditions B4-4
Manual Specification of the Minimum Gap Size B4-6
Switching off the Automatic Mesh Definition B4-9
Using the Local Initial Mesh Option B4-11
Specifying Control Planes B4-12
Creating a Second Local Initial Mesh B4-15

Advanced Examples

Application of EFD Zooming

Problem Statement C1-1
The EFD Zooming Approach to Solve the Problem C1-3
The Local Initial Mesh Approach. C1-12
Results C1-16

Textile Machine

Problem Statement C2-1
Opening the Model C2-2
Creating a FloEFD Project C2-3
Specifying Boundary Conditions C2-4
Specifying Rotating Walls C2-5
Specifying Initial Conditions C2-6
Specifying Goals. C2-7
Results (Smooth Walls) C2-8
Displaying Flow and Particles Trajectories C2-8
Modeling Rough Rotating Wall C2-10
Adjusting Wall Roughness C2-11
Results (Rough Walls) C2-11

Non-Newtonian Flow in a Channel with Cylinders

Problem Statement C3-1
Opening the Model C3-2
Defining Non-Newtonian Liquid C3-2
Project Definition C3-2
Specifying Boundary Conditions C3-3
Specifying Goals C3-4
Comparison with Water C3-4

Radiative Heat Transfer

Problem Statement C4-1
Opening the Model C4-2
Case 1: The reflector inner surface is a whitebody C4-3
Case 2: All reflector surfaces are blackbody C4-6
Case 3: The reflector is removed C4-7
Results C4-8

Rotating Impeller

Problem Statement C5-1
Opening the Model C5-2
Creating a FloEFD Project C5-2
Specifying Boundary Conditions C5-3
On Calculating the Impeller's Efficiency C5-5
Specifying Project Goals C5-5
Results C5-8

CPU Cooler

Problem Statement C6-1
Opening the Model C6-2
Creating a FloEFD Project C6-2
Adjusting the Computational Domain Size C6-3
Specifying the Rotating Region C6-3
Specifying Stationary Walls C6-5
Specifying Solid Materials C6-6
Specifying Heat Source C6-6
Specifying Initial Mesh Settings C6-6
Specifying Project Goals C6-9
Results C6-11

Oil Catch Can

Problem Statement C7-1
Opening the Model C7-2
Creating a FloEFD Project C7-2
Specifying Boundary Conditions C7-2
Specifying Project Goals C7-3
Setting Solution Adaptive Mesh Refinement C7-4
Defining Motor Oil Material C7-6
Studying the Motion of Oil Droplets C7-6
Results C7-8

Examples for HVAC Module

150W Halogen Floodlight

Problem Statement D1-1
Opening the Model D1-2
Creating a FloEFD Project D1-3
Adjusting the Computational Domain Size D1-3
Specifying Fluid Subdomain D1-4
Specifying Heat and Radiation Conditions D1-4
Specifying Solid Materials D1-8
Specifying Goals D1-9
Setting Local Initial Mesh D1-9
Adjusting the Calculation Control Options D1-10
Results D1-10

Hospital Room

Problem Statement D2-1
Model Configuration D2-2
Project Definition D2-3
Boundary Conditions D2-4
Specifying Heat Sources D2-5
Specifying Calculation Control Options D2-7
Specifying Goals D2-7
Adjusting Initial Mesh D2-7
Setting Local Initial Mesh D2-8
Results D2-9

Pollutant Dispersion in the Street Canyon

Problem Statement D3-1
Model Configuration D3-2
Project Definition D3-3
Adjusting the Computational Domain Size D3-3
Specifying Goals. D3-3
Setting Local Initial Mesh D3-4
Adjusting the Calculation Control Options D3-4
Specifying Tracer Study D3-5
Results D3-6

Examples for Electronics Cooling and LED Modules

Electronic Components

Problem Statement E1-1
Opening the Model E1-2
Creating a FloEFD Project E1-5
Specifying Boundary Conditions E1-6
Specifying Perforated Plates. E1-7
Specifying Two-Resistor Components E1-8
Specifying Heat Pipes. E1-10
Specifying Contact Resistances E1-11
Specifying Printed Circuit Board E1-12
Specifying Solid Materials E1-13
Specifying Project Goals E1-13
Adjusting the Initial Mesh E1-14
Specifying Local Initial Mesh Properties E1-15
Results E1-16

LED Lighting

Problem Statement E2-1
Opening the Model E2-3
Creating a FloEFD Project E2-4
Adjusting the Computational Domain Size E2-4
Specifying Solid Materials E2-5
Specifying Radiative Surfaces E2-7
Specifying LEDs E2-8
Specifying Goals E2-12
Results E2-12

Examples for Advanced CFD Module

Combustion in a Tube

Problem Statement F1-1
Case 1: Non-premixed combustion F1-2
Case 2: Premixed Combustion with ignition F1-9

Features List

This chapter contains the list of the physical and interface features of FloEFD as they appear in the tutorial examples. If you need to find an example of a certain feature or function usage, look for the desired feature in the left column and in its row you can see in which tutorial examples this feature is used. Usually, the first entrance of the feature in the tutorial contains the most detailed description. The tutorial examples are listed in Features List by their respective numbers. All tutorial examples are divided in three categories: First Steps, Intermediate and Advanced.

📖 *In the **First Steps** examples you will learn the basic principles of the FloEFD structure and interface.*

A1 - Ball Valve Design

A2 - Conjugate Heat Transfer

A3 - Porous Media

📖 *On the **Intermediate** level you will learn how to solve engineering problems with FloEFD, using some of the most common tasks as examples.*

B1 - Determination of Hydraulic Loss

B2 - Cylinder Drag Coefficient

B3 - Heat Exchanger Efficiency

B4 - Mesh Optimization

🕮 *In the **Advanced** examples you can see how to use a wide variety of the FloEFD features to solve real-life engineering problems. It is assumed that you successfully completed all First Steps examples before.*

C1 - Application of EFD Zooming

C2 - Textile Machine

C3 - Non-Newtonian Flow in a Channel with Cylinders

C4 - Radiative Heat Transfer

C5 - Rotating Impeller

C6 - CPU Cooler

C7 - Oil Catch Can

🕮 *In the examples for **HVAC Module** you can see how to use an additional capabilities of the FloEFD to solve Heating, Ventilation, and Air Conditioning tasks. This functionality is available for the HVAC module users only.*

D1 - 150W Halogen Floodlight

D2 - Hospital Room

D3 - Pollutant Dispersion in the Street Canyon

🕮 *In the examples for **Electronics Cooling Module** and **LED Module** you can see how to use an additional capabilities of the FloEFD to simulate a wide variety of electronic components. This functionality is available for the Electronics Cooling and LED modules users only.*

E1 - Electronic components

E2 - LED lighting

🕮 *In the examples for **Advanced CFD Module** you can see how to use an additional capabilities of the FloEFD to solve specific engineering problems, such as the combustion of gas-phase mixtures. This functionality is available for the Advanced CFD module users only.*

F1 - Combustion in a Tube

	First Steps			Intermediate				Advanced							Modules					
	A 1	A 2	A 3	B 1	B 2	B 3	B 4	C 1	C 2	C 3	C 4	C 5	C 6	C 7	D 1	D 2	D 3	E 1	E 2	F 1
DIMENSIONALITY																				
2D flow					✔															
3D flow	✔	✔	✔	✔		✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
ANALYSIS TYPE																				
External analysis					✔						✔		✔		✔		✔		✔	
Internal analysis	✔	✔	✔	✔		✔	✔	✔	✔	✔		✔		✔		✔		✔		✔
PHYSICAL FEATURES																				
Steady state analysis	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔		✔	✔	
Time-dependent (transient) analysis					✔												✔			✔
Liquids	✔			✔	✔	✔				✔										
Gases		✔	✔			✔	✔	✔	✔			✔	✔	✔	✔	✔	✔	✔	✔	
Non-Newtonian liquids										✔										
Combustible Mixtures																				✔
Multi-fluid analysis																				
Mixed flows							✔									✔				✔
Separated flows (as Fluid Subdomains)						✔									✔					
Heat conduction in solids		✔				✔		✔			✔		✔		✔			✔	✔	
Heat conduction in solids only											✔									
Gravitational effects							✔								✔	✔		✔	✔	
Laminar only flow										✔										
Porous media			✔				✔													

	First Steps			Intermediate				Advanced							Modules					
	A 1	A 2	A 3	B 1	B 2	B 3	B 4	C 1	C 2	C 3	C 4	C 5	C 6	C 7	D 1	D 2	D 3	E 1	E 2	F 1
Radiation											✔				✔				✔	
Absorption in solids															✔				✔	
Spectrum															✔				✔	
Roughness									✔											
Two-phase flows (fluid flows with particles or droplets)									✔					✔						
Rotation																				
Global rotating reference frame												✔								
Local rotating regions													✔							
CONDITIONS																				
Computational domain					✔			✔			✔		✔		✔		✔		✔	
Symmetry					✔	✔									✔		✔			
Initial and ambient conditions																				
Velocity parameters					✔												✔			
Dependency					✔												✔			
Thermodynamic parameters						✔							✔			✔				
Turbulence parameters					✔												✔			
Concentration							✔									✔				
Solid parameters													✔							

	First Steps			Intermediate				Advanced							Modules					
	A 1	A 2	A 3	B 1	B 2	B 3	B 4	C 1	C 2	C 3	C 4	C 5	C 6	C 7	D 1	D 2	D 3	E 1	E 2	F 1
Boundary conditions																				
Flow openings																				
Inlet mass flow	✔					✔			✔											✔
Inlet volume flow							✔			✔		✔		✔		✔				
Outlet volume flow							✔									✔				
Inlet velocity			✔	✔		✔														
Pressure openings																				
Static pressure	✔		✔	✔					✔	✔				✔						
Environment pressure		✔				✔	✔	✔				✔				✔		✔		✔
Wall																				
Real wall									✔			✔	✔							
Boundary condition parameters	✔		✔	✔		✔	✔		✔	✔		✔								
Transferred boundary conditions								✔												
Fans		✔						✔										✔	✔	
Contact resistances																		✔	✔	
Perforated plates																		✔	✔	
Volume conditions																				
Fluid Subdomain						✔									✔					
Initial conditions																				
Velocity parameters									✔											
Dependency									✔											

	First Steps			Intermediate				Advanced							Modules					
	A 1	A 2	A 3	B 1	B 2	B 3	B 4	C 1	C 2	C 3	C 4	C 5	C 6	C 7	D 1	D 2	D 3	E 1	E 2	F 1
Solid parameters											✔									
Solid material		✔				✔		✔			✔		✔		✔			✔	✔	
Semi-transparent															✔				✔	
Porous medium			✔				✔													
Heat sources																				
Surface sources																				
Heat generation rate								✔			✔					✔				
Volume sources																				
Temperature		✔													✔					
Heat generation rate		✔						✔					✔					✔	✔	✔
Goal-dependent sources															✔					
Radiative conditions																				
Radiation sources											✔				✔					
Radiative surfaces											✔				✔				✔	
Electronics module features (requires Electronics Cooling license)																				
Two-resistor components																		✔		
Heat pipe																		✔		
Printed circuit board																		✔		
LED module features (requires LED license)																				
LED components																			✔	

	First Steps			Intermediate				Advanced							Modules					
	A 1	A 2	A 3	B 1	B 2	B 3	B 4	C 1	C 2	C 3	C 4	C 5	C 6	C 7	D 1	D 2	D 3	E 1	E 2	F 1
Tracers (requires HVAC license)																				
Tracer Study Setting																	✔			
Surface sources																				
Mass flow																	✔			
PROJECT DEFINITION																				
Wizard and Navigator	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
From template					✔															
Clone project	✔		✔	✔	✔			✔	✔	✔	✔									✔
General settings					✔					✔										
Copy project's features								✔												
GOALS																				
Global goal		✔			✔				✔			✔	✔	✔		✔	✔		✔	✔
Surface goal	✔	✔	✔	✔				✔	✔	✔	✔	✔	✔	✔	✔		✔	✔	✔	✔
Volume goal		✔				✔		✔	✔		✔				✔			✔	✔	
Point goal													✔							
Equation goal			✔		✔					✔		✔	✔	✔						
MESH SETTINGS																				
Initial mesh																				
Automatic settings																				
Level of initial mesh								✔					✔		✔	✔	✔		✔	
Minimum gap size	✔	✔		✔			✔	✔	✔	✔		✔	✔			✔			✔	✔
Minimum wall thickness							✔				✔	✔				✔			✔	

	First Steps			Intermediate				Advanced							Modules					
	A 1	A 2	A 3	B 1	B 2	B 3	B 4	C 1	C 2	C 3	C 4	C 5	C 6	C 7	D 1	D 2	D 3	E 1	E 2	F 1
Manual adjustments																				
Control planes							✔						✔					✔		
Solid/fluid interface											✔		✔					✔		
Narrow channels							✔						✔							
Local initial mesh																				
Manual adjustments																				
Refining cells							✔								✔	✔	✔	✔		
Narrow channels							✔	✔									✔	✔		
TOOLS																				
Dependency					✔				✔						✔					
Custom units		✔						✔												
Engineering database																				
User-defined items		✔	✔					✔		✔				✔	✔		✔	✔	✔	✔
Check geometry				✔												✔				
Gasdynamic calculator				✔																
Toolbars						✔														
Filter																		✔		
Component control							✔	✔			✔	✔	✔			✔				
CALCULATION CONTROL OPTIONS																				
Finish conditions																	✔			✔
Result resolution level					✔			✔	✔			✔								
Solution adaptive mesh refinement														✔						

	First Steps			Intermediate				Advanced							Modules					
	A 1	A 2	A 3	B 1	B 2	B 3	B 4	C 1	C 2	C 3	C 4	C 5	C 6	C 7	D 1	D 2	D 3	E 1	E 2	F 1
Calculate comfort parameters																✔				
RUNNING CALCULATION																				
Batch run					✔			✔			✔									
MONITORING CALCULATION																				
Goal plot	✔																			
Preview	✔																			
GETTING RESULTS																				
Cut plot	✔	✔		✔		✔	✔	✔	✔						✔	✔	✔		✔	✔
Surface plot	✔	✔																	✔	
Isosurfaces	✔															✔	✔			
Flow trajectories	✔	✔	✔			✔			✔								✔			
Particle study									✔					✔						
XY plot	✔																			
Surface parameters	✔					✔														
Volume parameters																✔				
Goal plot		✔	✔	✔	✔	✔		✔		✔			✔					✔		
Display parameters				✔																
Results summary																✔				
Callouts																			✔	

	First Steps			Intermediate				Advanced							Modules					
	A 1	A 2	A 3	B 1	B 2	B 3	B 4	C 1	C 2	C 3	C 4	C 5	C 6	C 7	D 1	D 2	D 3	E 1	E 2	F 1
Display mode																				
Show/Hide model geometry				✔		✔	✔													
Transparency	✔		✔																	
Apply lighting	✔																			
OPTIONS																				
Use CAD geometry							✔													
Display mesh							✔													

A

First Steps

The **First Steps** examples presented below demonstrate the basic principles of the FloEFD structure and interface. Its strongly recommended to complete these tutorials examples first.

A1 - Ball Valve Design

A2 - Conjugate Heat Transfer

A3 - Porous Media

A1

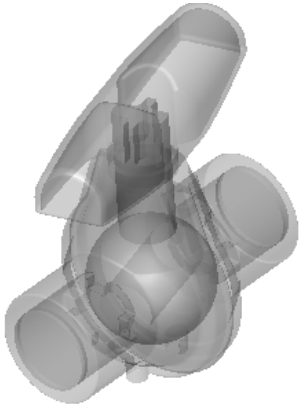
Ball Valve Design

This tutorial deals with the flow of water through a ball valve assembly before and after some design changes. The objective is to show how easy fluid flow simulation can be with FloEFD and how simple it is to analyze design variations. These two factors make FloEFD the perfect tool for engineers who want to test the impact of their design changes.

Opening the Model

1 Copy the **A1 - Ball Valve** folder from the installation directory (**<install_dir>\Examples\Tutorial Examples**) into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files. Run **FloEFD**.

2 Click **File > Open**. In the **File Open** dialog box, browse to the `ball_valve.asm` assembly located in the **A1 - Ball Valve** folder and click **Open** (or double-click the assembly). Alternatively, you can drag and drop the `ball_valve.asm` file to an empty area of Creo Elements/Pro & Creo Parametric window.



📖 *This is a ball valve. Turning the handle closes or opens the valve. The assembly mate angle controls the opening angle.*

3 Highlight the **lids** by clicking the components in the Creo Elements/Pro & Creo Parametric Model Tree (LID_1 and LID_2).

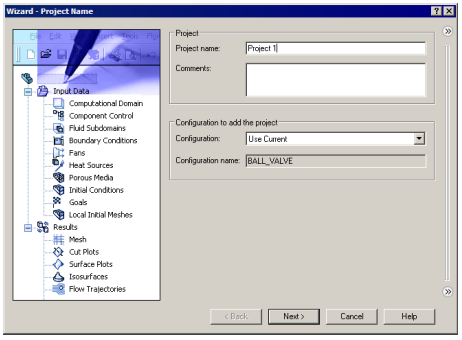
📖 *We utilize this model for the FloEFD simulation without any significant changes. The user simply closes the interior volume using extrusions that we call lids. In this example the lids are made semi-transparent so you may look into the valve.*

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the* `ball_valve.asm` *assembly located in the* ***A1 - Ball Valve\Ready To Run*** *folder and run the desired projects.*

Creating a FloEFD Project

1 In the main menu click **Flow Analysis > Project > Wizard**.

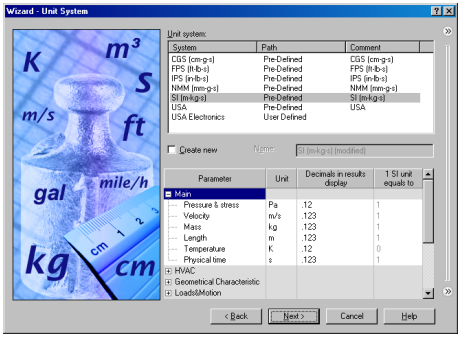
2 Once inside the Wizard, type a new FloEFD project name: `Project 1`.



📖 *FloEFD will create a new project and store all data in a new folder.*

Click **Next**.

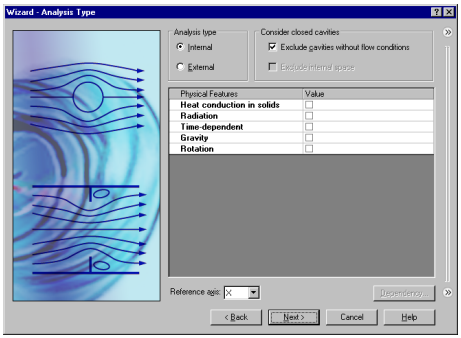
3 Choose the system of units (**SI** for this project). Please keep in mind that after finishing the Wizard you can change the unit system at any time by clicking **Flow Analysis > Units**.



📖 *Within FloEFD, there are several predefined systems of units. You can also define your own and switch between them at any time.*

Click **Next**.

4 Keep the default **Internal** analysis type. Do not include any physical features.

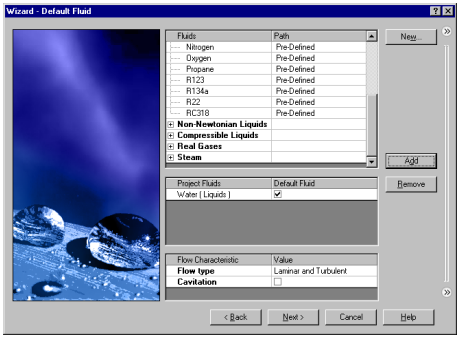


🕮 *We want to analyze the flow* **through** *the structure. This is what we call an internal analysis. The alternative is an external analysis, which is the flow* **around** *an object. In this dialog box you can also choose to ignore cavities that are not relevant to the flow analysis, so that FloEFD will not waste memory and CPU resources to take them into account.*

🕮 *Not only will FloEFD calculate the fluid flow, but can also take into account heat conduction within the solid, including surface-to-surface radiation. Transient (time-dependent) analyses are also possible. Gravitational effects can be included for natural convection cases. Analysis of rotating equipment is one more option available. We skip all these features, as none of them is needed in this simple example.*

Click **Next**.

5 In the **Fluids** tree expand the **Liquids** item and choose **Water** as the fluid. You can either double-click **Water** or select the item in the tree and click **Add**.



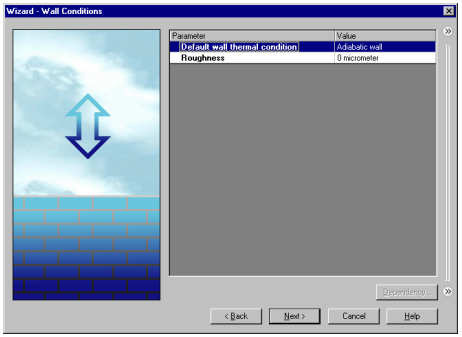
🕮 *FloEFD is capable of calculating flow of fluids of different types in the same analysis, but fluids of different types must be separated by walls. A mixing of fluids may be considered only if the fluids are of the same type.*

🕮 *FloEFD has an integrated database containing properties of several liquids, gases and solids. Solids are used in conjugate heat conduction analyses. You can easily create your own materials. Up to ten liquids or gases can be chosen for each analysis run.*

🕮 *FloEFD can analyze any flow type: Turbulent only, Laminar only or Laminar and Turbulent. The turbulent equations can be disregarded if the flow is entirely laminar. FloEFD can also handle low and high Mach number compressible flows for gases. For this demonstration we will perform a fluid flow simulation using a liquid and will keep the default flow characteristics.*

Click **Next**.

6 Click **Next** accepting the default wall conditions.

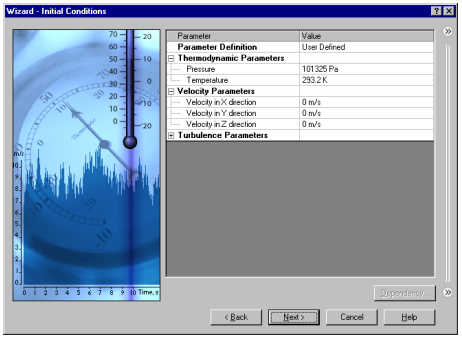


🕮 *Since we did not choose to consider heat conduction in solids, we have an option to define a value of heat transfer for all surfaces of the model being in contact with the fluid. Keep the default* **Adiabatic wall** *to specify that the walls are perfectly insulated.*

🕮 *You can also specify a wall roughness value applied by default to all model walls. The specified roughness value is the* R_z *value.*

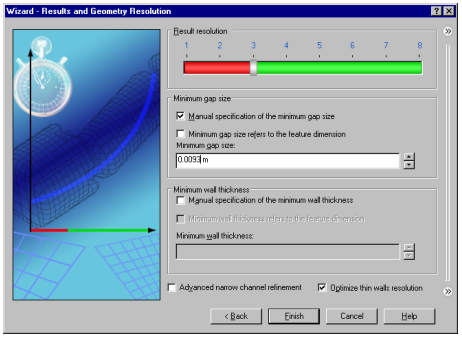
🕮 *To set a heat transfer or roughness value for a specific wall, you can define a* ***Real Wall*** *boundary condition.*

7 Click **Next** accepting the default for the initial conditions.



🕮 *On this step we can change the default settings for pressure, temperature and velocity. The closer these values to the final values determined in the analysis, the quicker the analysis will finish. Since we do not have any knowledge of the expected final values, we will not modify them for this demonstration.*

8 Accept the default for the **Result Resolution**.



🕮 *Result Resolution is a measure of the desired level of accuracy of the results. It controls not only the resolution of the geometry by the mesh, but also sets many parameters for the solver, e.g. convergence criteria. The higher the Result Resolution, the finer the mesh will be and the stricter convergence criteria will be set. Thus, Result Resolution determines the balance between results precision and computation time. Entering values for the minimum gap size and minimum wall thickness is important when you*

have small features. Accurately setting these values ensures that the small features of the model will not be "passed over" by the mesh. For our model we type the value of the minimum flow passage as the minimum gap size.

Select the **Manual specification of the minimum gap size** check box. Type the value of `0.0093 m` for the **Minimum gap size**.

Click **Finish**.

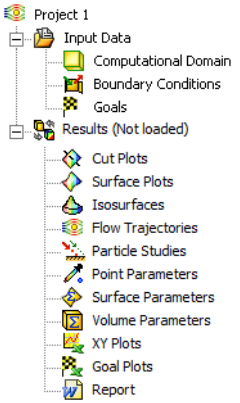
Now FloEFD creates a new project with the FloEFD data attached.

The FloEFD Projects tree and the FloEFD Analysis tree appears in the FloEFD Analysis tab of the Navigator panel.

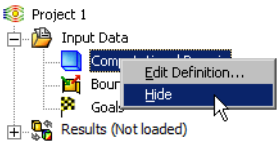
Go to the **FloEFD Analysis** tab and expand all the items in the FloEFD Analysis tree.

Click to hide the FloEFD projects tree.

📖 *We will use the FloEFD Analysis tree to define our analysis, just as you use the Model Tree to design your models. The FloEFD analysis tree is fully customizable; anytime you can select which folders are shown and which folders are hidden. A hidden folder becomes visible when you add a new feature of the corresponding type. The folder remains visible until the last feature of this type is deleted.*



Right-click the **Computational Domain** icon and select **Hide** to hide the wireframe box.

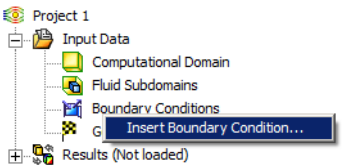


The Computational Domain icon is used to modify the size of the volume being analyzed. The wireframe box enveloping the model is the visualization of the limits of the computational domain.

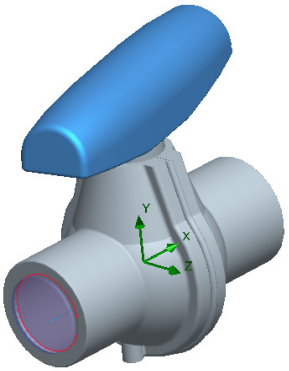
Specifying Boundary Conditions

A **boundary condition** is required where fluid enters or exits the model and can be specified as a Pressure, Mass Flow Rate, Volume Flow Rate or Velocity.

1 In the FloEFD Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

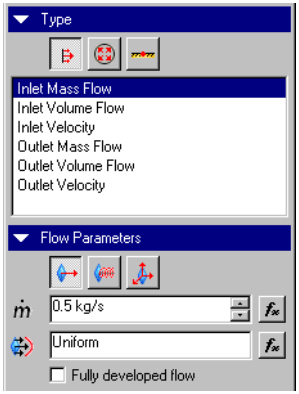


2 Select the **inner** face of the **LID_1** part as shown. (To access the inner face, set Filter to **Geometry** and right-click the **Lid_1** until the inner face is highlighted, then click the left mouse button).



3 Select **Flow Openings** and **Inlet Mass Flow**.

4 Set the **Mass Flow Rate** to `0.5 kg/s`.

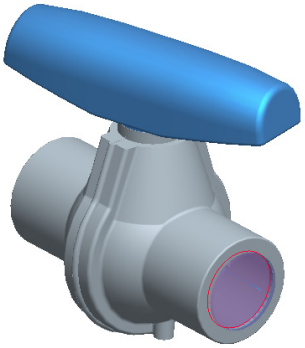


5 Click **OK**. The new **Inlet Mass Flow 1** item appears in the FloEFD Analysis tree.

With the definition just made, we told FloEFD that at this opening 0.5 kilogram of water per second is flowing into the valve. Within this dialog we can also specify swirling of the flow, a non-uniform profile and time-dependent properties of the flow. The mass flow rate at the outlet does not need to be specified due to the conservation of mass; inlet mass flow rate equals outlet mass flow rate. Therefore, a different condition must be specified, such as outlet pressure.

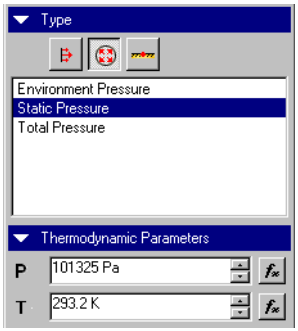
6 Select the **inner** face of the **LID_2** part as shown.

7 In the FloEFD Analysis Tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

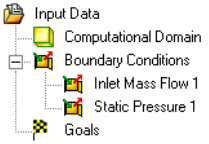


8 Select **Pressure Openings** and **Static Pressure**.

9 Keep the defaults under **Thermodynamic Parameters**, **Turbulence Parameters**, **Boundary Layer** and **Options**.



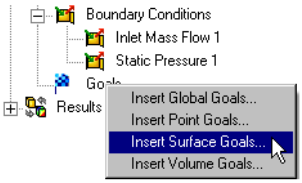
10 Click **OK**. The new **Static Pressure 1** item appears in the FloEFD Analysis tree.



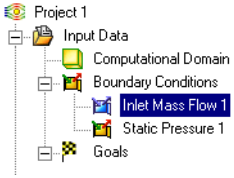
📖 *With the definition just made, we told FloEFD that at this opening the fluid exits the model to an area of static atmospheric pressure. Within this dialog we can also set a time-dependent properties pressure.*

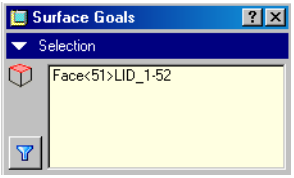
Specifying the Engineering Goal

1 Right-click the **Goals** icon in the FloEFD Analysis tree and select **Insert Surface Goals**.

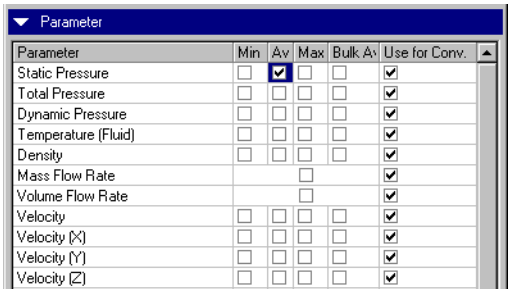


2 Click the **Inlet Mass Flow 1** item to select the face where the goal is going to be applied.



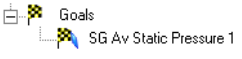


3 In the **Parameter** table, select the **Av** check box in the **Static Pressure** row. The already selected **Use for Conv.** check box means that the created goal will be used for convergence control.



🕮 *If the* **Use for Conv.** *(Use for Convergence Control) check box is not selected, the goal will not influence the calculation stopping criteria. Such goals can be used as monitoring parameters to give you additional information about processes in your model without influencing the other results and the total calculation time.*

4 Click **OK** . The new **SG Av Static Pressure 1** item appears in the FloEFD Analysis tree.

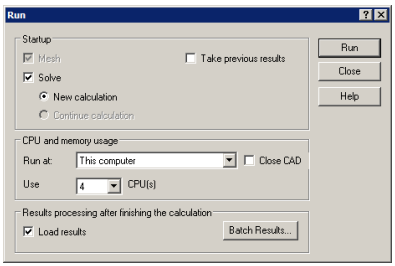


🕮 *Engineering goals are the parameters of interest. Setting goals is a way of conveying to FloEFD what you are trying to get out of the analysis, as well as a way to reduce the time FloEFD needs to reach a solution. By setting a parameter as a project goal you give FloEFD information about parameters that are important to converge upon (the parameters selected as goals) and parameters that can be computed with less accuracy (the parameters not selected as goals) in the interest of the calculation time. Goals can be set throughout the entire domain (Global Goals), within a selected volume (Volume Goals), for a selected surface area (Surface Goals), or at given point (Point Goals). Furthermore, FloEFD can consider the average value, the minimum value or the maximum value of the goal. You can also define an Equation Goal that is a goal defined by an equation involving basic mathematical functions with existing goals and input data parameters as variables. The equation goal allows you to calculate the parameter of interest (i.e., pressure drop) and keeps this information in the project for later reference.*

Click **File > Save**.

Running the Calculation

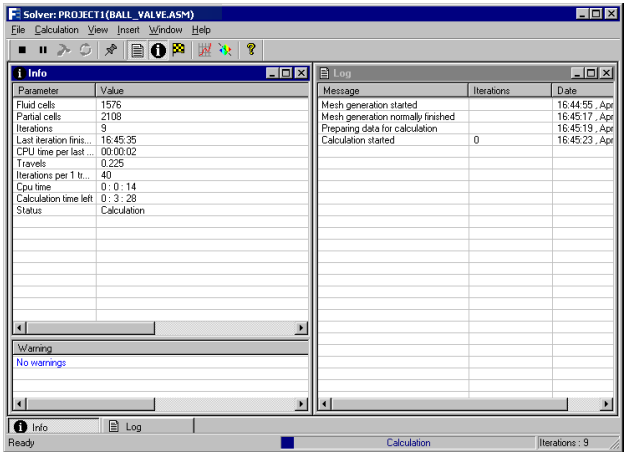
1 Click **Flow Analysis > Solve > Run**.



🕮 *The already selected* **Load results** *check box means that the results will be automatically loaded after finishing the calculation.*

2 Click **Run**.

🕮 *The solver takes less than a minute to run on a typical PC.*



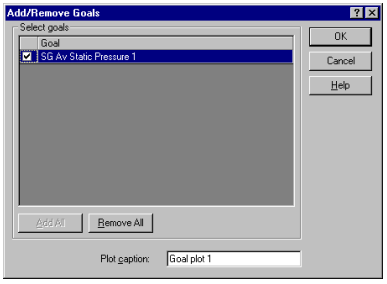
📖 *This is the solver monitor dialog box. By default, on the left is a log of each step taken in the solution process. On the right is the information dialog box with mesh information and warnings concerning the analysis. Do not be surprised when the error message "A vortex crosses the pressure opening" appears. We will explain this later during the demonstration.*

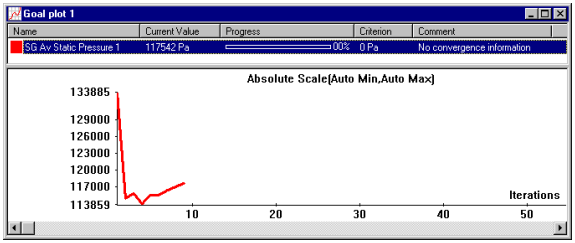
1 After the calculation has started and several first iterations has passed (keep your eye on the **Iterations** line in the **Info** window), click the **Suspend** button on the **Solver** toolbar.

📖 *We employ the Suspend option only due to extreme simplicity of the current example, which otherwise could be calculated too fast, leaving you not enough time to perform the subsequent steps of monitoring. Normally you can use the monitoring tools without suspending the calculation.*

2 Click **Insert Goal Plot** on the **Solver** toolbar. The **Add/Remove Goals** dialog box appears.

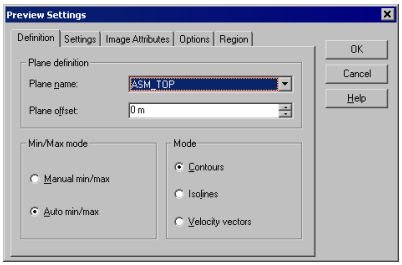
3 Select the **SG Average Static Pressure 1** in the **Select goals** list and click **OK**.





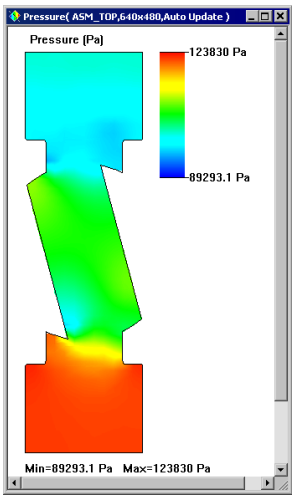
This is the Goals dialog box and each goal created earlier is listed in the table at top. Here you can see the current value and graph for each goal as well as the current progress towards completion given as a percentage. The progress value is only an estimate and the rate of progress generally increases with time.

4 Click **Insert Preview** on the **Solver** toolbar. The **Preview Settings** dialog box will appear.



5 To create a preview plot, you can select any plane from the **Plane name** list and then press **OK**. For this model, `ASM_TOP` can be a good choice.

The preview allows you to look at the results while the calculation is still running. This helps to determine if all the boundary conditions are correctly defined and gives the user an idea of how the solution will look even at this early stage. At the start of the run the results might look odd or change abruptly. However, as the run progresses these changes will lessen and the results will settle in on a converged solution. The result can be displayed either in contour-, isoline- or vector-representation.



6 Click the **Suspend** button again to let the solver go on.

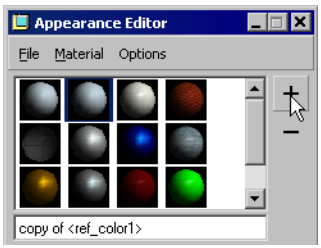
7 When the solver is finished, close the monitor by clicking **File > Close**.

Adjusting Model Transparency

Click **View > Color and Appearance**.

In the **Appearance Editor** click the plus button to add a new appearance.

Go to **Advanced** tab and adjust transparency to about 80.



Click **Apply** and then **Close**.

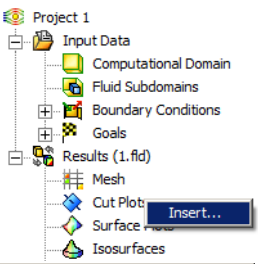
🕮 *The first step for results processing is to create a transparent view of the geometry, a 'glass-body'. This way you can easily see where cut planes etc. are located with respect to the geometry.*

✎ *In Creo Parametric click* ***View > Model Display > Appearance Gallery > More Appearances****. In the* ***Appearance Editor*** *adjust transparency to about 80.*

Viewing Cut Plots

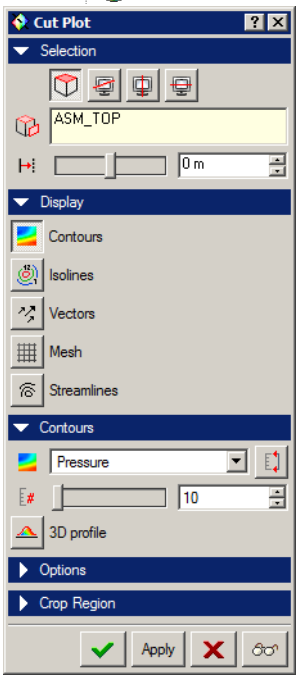
A cut plot displays the distribution of the selected parameter on a certain plane. It can be represented as a contour plot, isolines, vectors, or as arbitrary combination of the above (e.g. contours with overlaid vectors).

1 In the FloEFD Analysis tree, right-click the **Cut Plots** icon and select **Insert**.

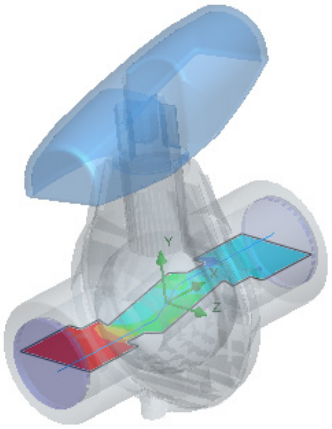


2 Go to the Creo Elements/Pro & Creo Parametric **Model Tree** tab, click **Settings > Tree Filters**, make sure that **Features** is checked, click **OK** and select **ASM_TOP** item in the Model Tree.

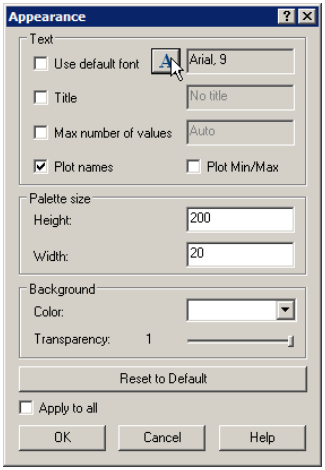
3 Click **OK** .



You will see the plot like the one shown below.

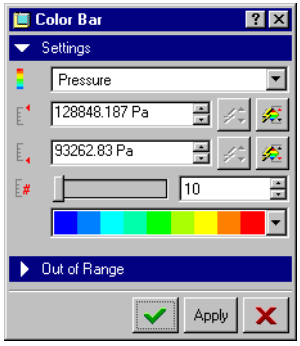


🕮 *If you want to set up the color bar display options (change the color bar font, background, etc.), right-click the color bar and select* **Appearance**. *To change the color bar font, deselect* **Use default font** *and click* A. *Then in the font dialog, select the desired font, its size and color and click* **OK**. *Click* **OK** *in the appearance dialog.*

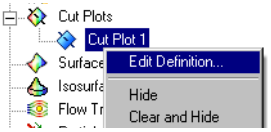


🕮 *You can to set up the callout display options in the same way as the color bar appearance.*

🕮 *If you want to access additional options for this and other plots, you can double-click on the color bar. Some options available here include changing the displayed parameter as well as changing the min/max plot values. The best way to learn each of these options is thorough experimentation.*



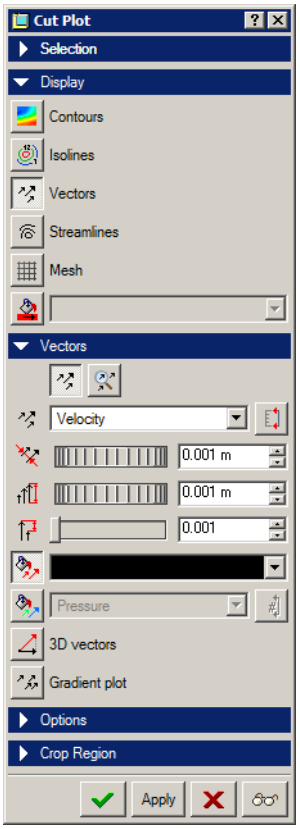
4 Change the contour cut plot to a vector cut plot. To do this, right-click the **Cut Plot 1** icon and select **Edit Definition**.



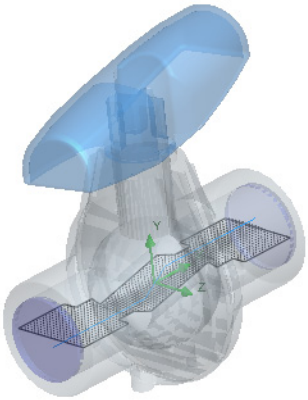
5 Under **Display**, clear **Contours** and select **Vectors** .

The vectors size and spacing can be controlled under the **Vectors**.

6 Click **OK** .



You will see the plot like the one shown below.

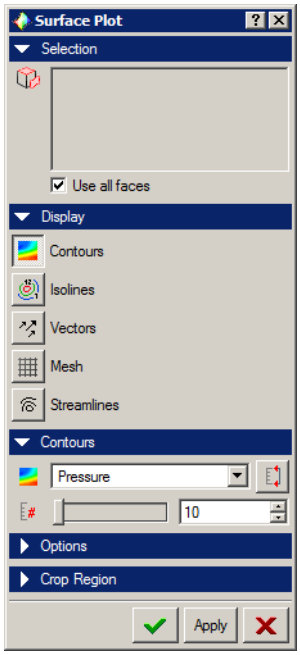


Viewing Surface Plots

Right-click the **Cut Plot 1** icon and select **Hide**.

1 Right-click the **Surface Plots** icon and select **Insert**.

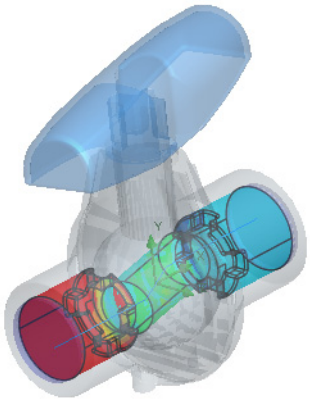
2 Select the **Use all faces** check box.



🕮 *The same basic options are available for Surface Plots as for Cut Plots. Feel free to experiment with different combinations on your own.*

3 Click **OK**.

You will see the plot like the one shown below.



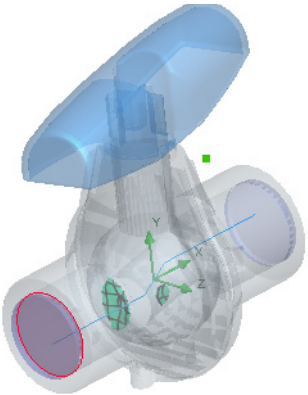
🕮 *This plot shows the pressure (or other parameter selected) distribution on all faces of the valve in contact with the fluid. You can also select one or more single surfaces for this plot, which do not have to be planar.*

Right-click the **Surface Plot 1** icon and select **Hide**.

1 Right-click the **Isosurfaces** icon and select **Insert**.

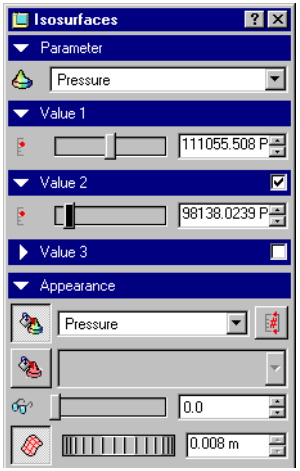
2 Keep the default value under **Value 1**.

3 Under **Appearance**, select **Grid** and click **OK**. You will see the isosurface like the one show in the picture right.



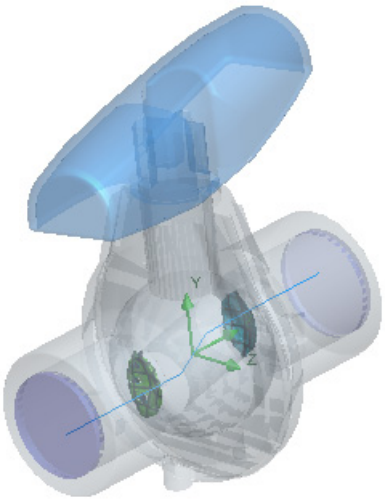
📖 *The Isosurface is a 3-Dimensional surface created by FloEFD at a constant value for a specific variable.*

4 Right-click the **Isosurface 1** icon and select **Edit Definition**. Enable **Value 2** and specify some value in the appeared box that is different to the **Value 1**.



5 Click **OK**.

You will see the isosurfaces like the ones shown below.



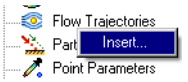
📖 *The isosurface is a useful way of determining the exact 3D area, where the flow reaches a certain value of pressure, velocity or other parameter.*

Viewing Flow Trajectories

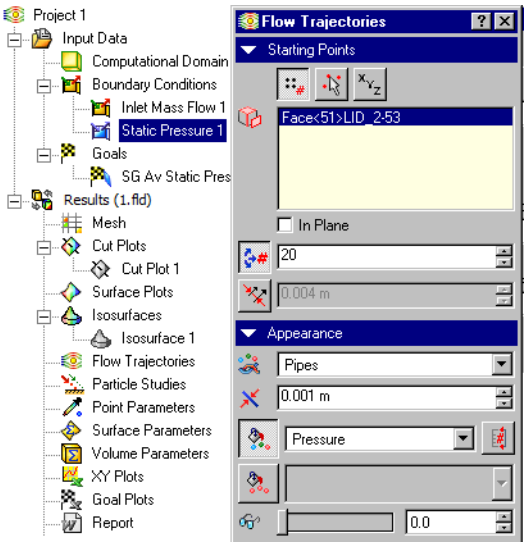
Using Flow trajectories you can show the flow streamlines. Flow trajectories provide a very good image of the 3D fluid flow. You can also see how parameters change along each trajectory by exporting data into Microsoft® Excel®.

1 Right-click the **Isosurfaces** icon and select **Hide**.

2 Right-click the **Flow Trajectories** icon and select **Insert**.



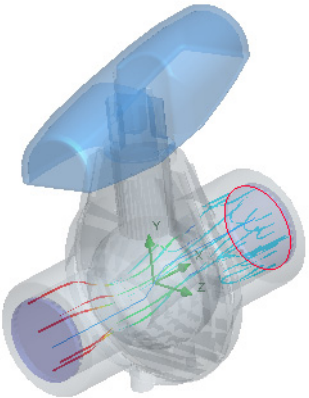
3 In the FloEFD Analysis tab click the **Static Pressure 1** item to select the inner face of the **LID_2**.



4 Set the **Number of Points** to `16`.

5 Under **Appearance**, set **Draw Trajectories as** **Bands**.

6 Click **OK** .You will the flow trajectories like the ones shown in the picture right.



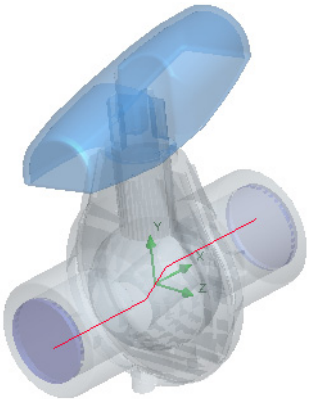
For this plot we selected the outlet lid (any flat face or sketch can be selected) and therefore every trajectory crosses that selected face. Notice the trajectories that are entering and exiting through the exit lid. This is the reason for the warning we received during the calculation. FloEFD warns us of inappropriate analysis conditions so that we do not need to be CFD experts. When flow both enters and exits the same opening, the accuracy of the results will worsen. In a case like this, one would typically add the next component to the model (say, a pipe extending the computational domain) so that the vortex does not occur at opening.

Viewing XY Plots

Right-click the **Flow Trajectories 1** icon and select **Hide**.

We will plot velocity and pressure distributions along the valve using the already created Creo Elements/Pro & Creo Parametric sketch containing several lines.

This sketch work does not have to be done ahead of time and your sketch lines can be created after the calculation is finished. Take a look at Sketch1 in the Model Tree.



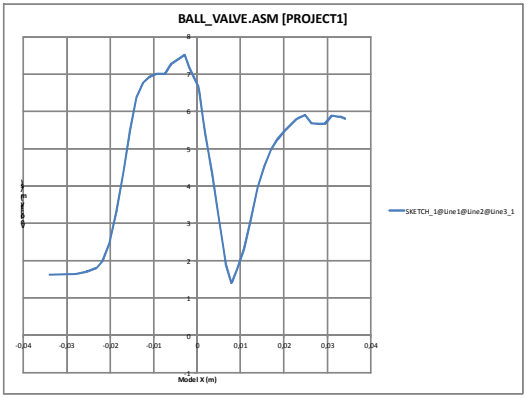
1 Right-click the **XY Plots** icon and select **Insert**.

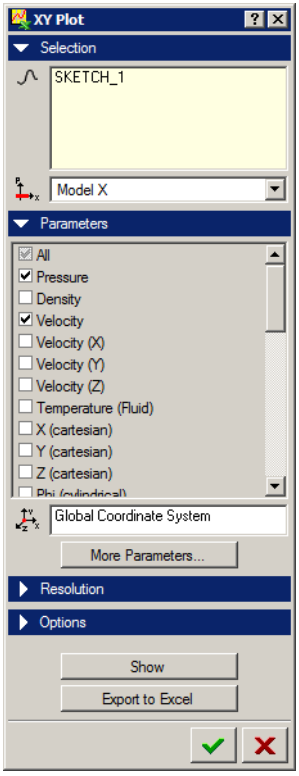
2 Choose **Velocity** and **Pressure** as **Parameters**. Select **Sketch1** from the Model tree.

3 Select **Model X** as abscissa and leave all other options default.

Leave all other options as defaults.

4 Click **Export to Excel**. Excel will open and generate two columns of data points together with two charts for Velocity and for Pressure, respectively. One of these charts is shown below. You will need to toggle between different sheets in Excel to view each chart.





5 Click **OK**.

The XY Plot allows you to view any result along sketched lines. The data is put directly into Excel or displayed as images in the bottom pane.

Viewing Surface Parameters

Surface Parameters is a feature used to determine the values of pressure, forces, heat fluxes as well as many other variables on any face in your model contacting the fluid. For this type of analysis, a calculation of the average static pressure drop from the valve inlet to outlet would probably be of some interest.

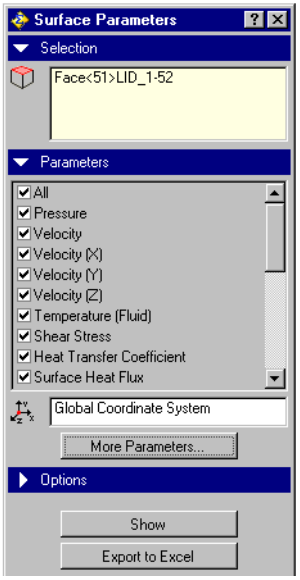
1 Right-click the **Surface Parameters** icon and select **Insert**.

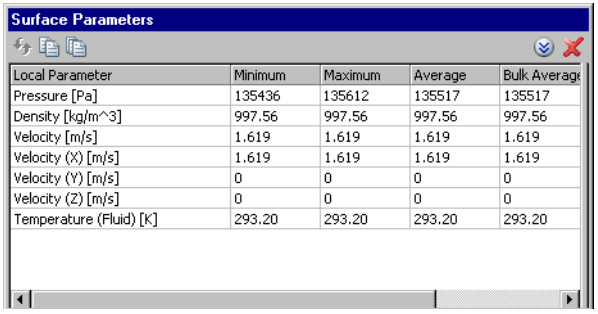
2 In the FloEFD Analysis tab click the **Inlet Mass Flow 1** item to select the inner face of the **LID_1**.

3 Under **Parameters**, select **All**.

4 Click **Show**.

5 Take a look at the local parameters.





Local Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	135436	135612	135517	135517
Density [kg/m^3]	997.56	997.56	997.56	997.56
Velocity [m/s]	1.619	1.619	1.619	1.619
Velocity (X) [m/s]	1.619	1.619	1.619	1.619
Velocity (Y) [m/s]	0	0	0	0
Velocity (Z) [m/s]	0	0	0	0
Temperature (Fluid) [K]	293.20	293.20	293.20	293.20

The average static pressure at the inlet face is shown to be about 135500 Pa. We already know that the outlet static pressure is 101325 Pa since we have specified it previously as a boundary condition. So, the average static pressure drop through the valve is about 34000 Pa.

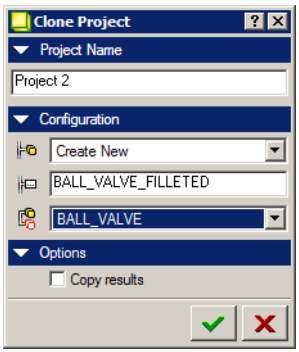
6 Close the **Surface Parameters** dialog box.

Analyzing a Design Variant in the Creo Elements/Pro & Creo Parametric Ball part

📖 *This section is intended to show you how easy it is to analyze design variations. The variations can be different geometric dimensions, new features, new parts in an assembly – whatever! This is the heart of FloEFD and this allows design engineers to quickly and easily determine which designs have promise, and which designs are unlikely to be successful. For this example, we will see how filleting two sharp edges will influence the pressure drop through the valve. If there is no improvement, it will not be worth the extra manufacturing costs.*

Cloning the Project

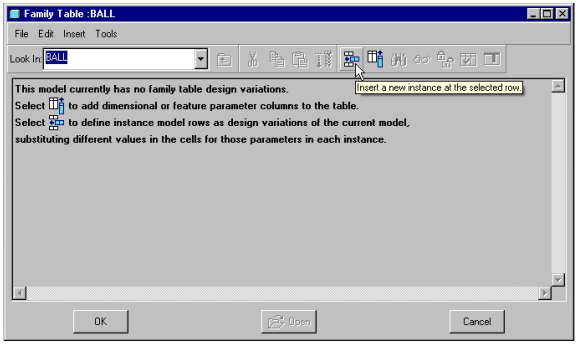
1. Click **Flow Analysis > Project > Clone Project**.
2. In the **Project Name**, type `Project 2`.
3. In the **Configuration to Add the Project** select **Create new**.
4. In the **Configuration Name** list, type `BALL_VALVE_FILLETED`.Click **OK**.



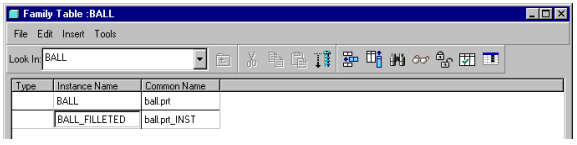
Now the FloEFD project we have chosen is added to the project which contains the geometry that has been changed. All our input data are copied, so we do not need to define our openings or goals again. The Boundary Conditions can be changed, deleted or added.

Creating a New Instance in the Family Table

1. In the Model Tree, right-click the **BALL** icon and select **Open**. A new window opens with the BALL part.
2. Click **Tools > Family Table**.
3. In the **Family Table** dialog box click **Insert a new instance at the selected row** .

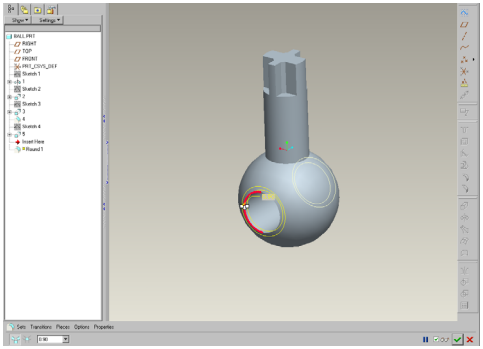

Family Table :BALL
File Edit Insert Tools
Look In: BALL
Insert a new instance at the selected row.
This model currently has no family table design variations.
Select to add dimensional or feature parameter columns to the table.
Select to define instance model rows as design variations of the current model,
substituting different values in the cells for those parameters in each instance.
OK
Open
Cancel

4. Change the name of the new configuration from **BALL_INST** to the more clear name `BALL_FILLETED`.
5. Click **Open**. A new window showing the new instance **BALL_FILLETED** appears.

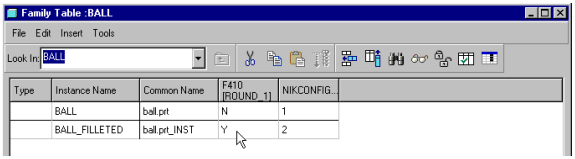


Changing the Model Geometry

1. Select sharp edges and make them round with radius of 0.9 mm.
2. Click **File > Save** to save the **BALL** part with the new instance containing the changed geometry.
3. Click **File > Close Window**. to close the window with the **BALL_FILLETED** instance.



4. The window with the **BALL** part in generic configuration is still opened. Click **Tools > Family Table** to check that the new configuration with rounded edges is added to the Family Table of the **BALL** part.

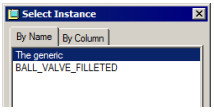


5. Click **File > Close Window** to close the window with the **BALL** part.

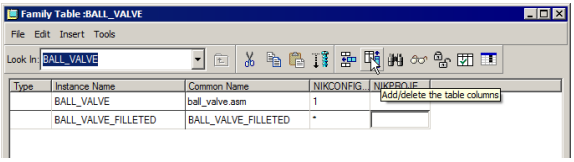
Replacing the Instance in the Assembly

1. In the still opened **BALL_VALVE_FILLETED** window click **File** > **Open** and select the **In Session** option .

1. Open the **ball_valve.asm** assembly and select the **generic** instance.

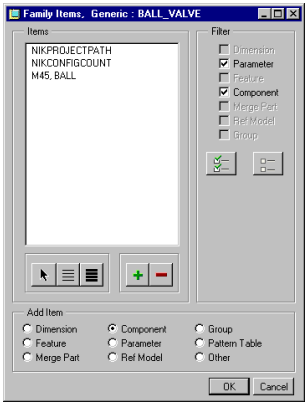


2. Click **Tools > Family Table**.
3. In the **Family Table** dialog box click **Add/delete the table columns** .



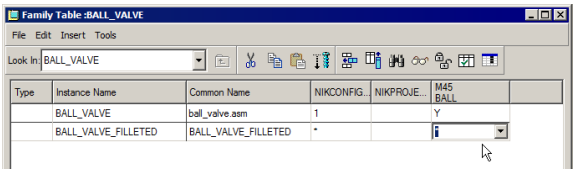
4 In the **Family Items** dialog box select **Component** under **Add Item** and then select **BALL.PRT** in the Model Tree.

5 Click **OK**. The new **M45 BALL** column is added to the family table.



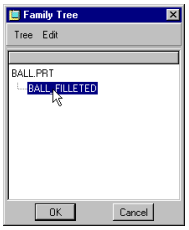
📖 *In the new column you can choose between **Y**, **N** and ***** for each of the instances. Selecting **N** will suppress the instance and all the differences and changes it contains, selecting **Y** unsuppress the instance and ***** leaves the instance in the same state as it is defined for the whole column in the column's top cell.*

6 In the **Family Table** dialog box select the cell of the new **M45 BALL** column belonging to the **BALL_VALVE_FILLETED** row.

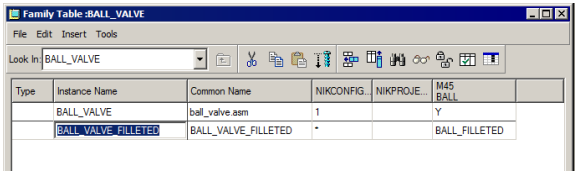


7 Click **Tools > Replace Using > Family Member** and select the **BALL_FILLETED** instance containing the changed geometry with the rounded edges.

8 Click **OK**.



9 In the **Family Table** dialog box select the **BALL_VALVE_FILLETED** row and click **Open**. The **BALL_VALVE_FILLETED** window opens and the **BALL** part now has rounded edges in this project.



Since all conditions have been copied into the new instance you can start calculation immediately. Please follow the previously described steps for solving and viewing the results.

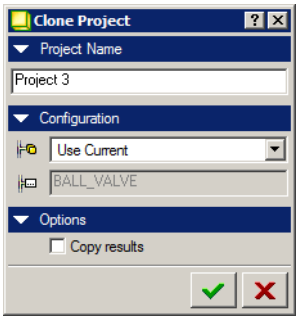
Analyzing a Design Variant in the FloEFD Application

🕮 *In the previous sections we examined how you could compare results from different geometries. You may also want to run the same geometry over a range of flow rates. This section shows how quick and easy it can be to do that kind of parametric study. Here we are going to change the mass flow to 0.75 kg/s.*

Activate the **Project 1** in the FloEFD Projects tree.

1 Create a copy of the **Project 1** project by clicking **Flow Analysis > Project > Clone Project**.

2 Type `Project 3` for the new project name and click **OK**.



FloEFD now creates a new project. All our input data are copied, so we do not need to define our openings or goals again. The Boundary Conditions can be changed, deleted or added. All changes to the geometry will only be applied to this new configuration, so the old results remain valid. After changing the inlet flow rate value to 0.75 kg/s you would be ready to run again. Please follow the previously described steps for solving and for viewing the results.

Imagine being the designer of this ball valve. How would you make decisions concerning your design? If you had to determine whether the benefit of modifying the design as we have just done outweighted the extra costs, how would you do this? Engineers have to make decisions such as this every day, and FloEFD is a tool to help them make those decisions. Every engineer who is required to make design decisions involving fluid and heat transfer should use FloEFD to test their ideas, allowing for fewer prototypes and quicker design cycles.

A2

Conjugate Heat Transfer

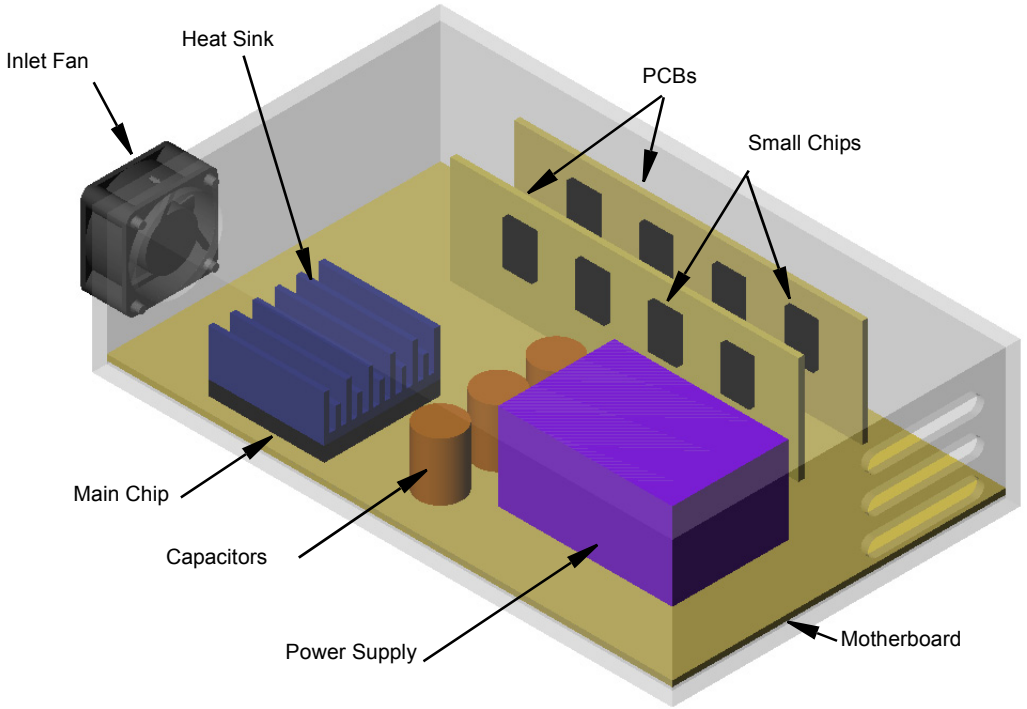
This tutorial covers the basic steps required to set up a flow analysis problem including heat conduction in solids. This example is particularly pertinent to users interested in analyzing flow and heat conduction within electronics devices, although the basic principles are applicable to all thermal problems. It is assumed that you have already completed the *Ball Valve Design* tutorial since it teaches the basic principles of using FloEFD in greater detail.

Opening the Model

1 Copy the **A2 - Conjugate Heat Transfer** folder into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files. Run **FloEFD**. Click **File > Open**.

📖 In the **File Open** dialog box, browse to the `enclosure_assembly.asm` assembly located in the **A2 - Conjugate Heat Transfer** folder and click **Open**.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the* `enclosure_assembly.asm` *assembly located in the* ***A2 - Conjugate Heat Transfer\Ready To Run*** *folder and run the project.*

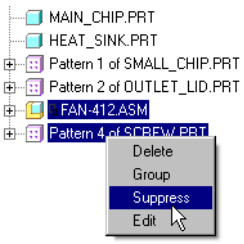


Preparing the Model

In a typical assembly there may be many features, parts or sub-assemblies that are not necessary for the analysis. Prior to creating a FloEFD project, it is a good practice to check the model to find components that can be removed from the analysis. Excluding these components reduces the computer resources and calculation time required for the analysis. The assembly consists of the following components: enclosure, motherboard and two smaller PCBs, capacitors, power supply, heat sink, chips, fan, screws, fan housing, and lids. You can highlight these components by clicking them in the Creo Elements/Pro & Creo Parametric Model Tree. In this tutorial we will simulate the fan by specifying a **Fan** boundary condition on the inner face of the inlet lid. The fan has a very complex geometry that may cause delays while rebuilding the model. Since it is outside the enclosure, we can exclude it by suppressing it.

1 In the Model Tree, select the **FAN-412** subassembly and **Pattern 4 of SCREW** item.

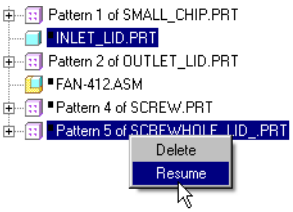
2 Right-click any of the selected items and select **Suppress**. Then click **OK** to confirm suppressing.
Suppressing fan and its screws leaves open five holes in the enclosure. Since we are going to perform an internal analysis, all the holes must be closed with lids.



To save your time, we created the lids and included them in this model. You just need to unsuppress them. Please make sure that the Tree Filters settings allow you to see the supressed objects in the Model Tree.

3 In the Model tree, select the **INLET_LID** and **Pattern 5 of SCREWHOLE_LID_** items .

4 Right-click any of the selected items and select **Resume**.

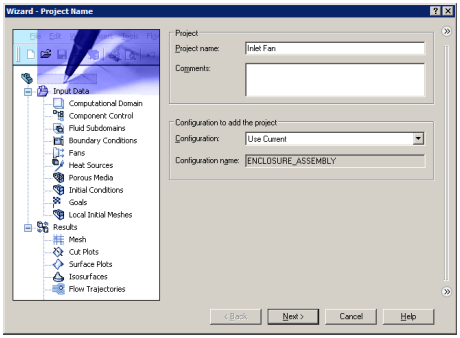


Now you can start with FloEFD.

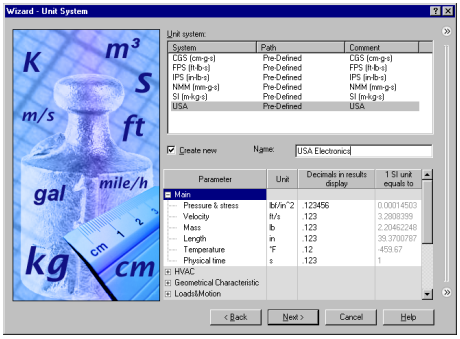
Creating a FloEFD Project

1 Click **Flow Analysis > Project > Wizard**.

2 Once inside the **Wizard**, type a new FloEFD project name: `Inlet Fan`. Click **Next**.
Now we will create a new system of units named **USA Electronics** that is better suited for our analysis.



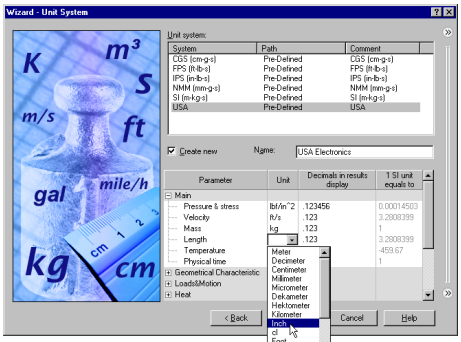
3 In the **Unit system** list select the **USA** system of units. Select **Create new** to add a new system of units to the Engineering Database and name it `USA Electronics`.



📖 *FloEFD allows you to work with several pre-defined unit systems but often it is more convenient to define your own custom unit system. Both pre-defined and custom unit systems are stored in the* ***Engineering Database****. You can create the desired system of units in the* ***Engineering Database*** *or in the* ***Wizard****.*

📖 *By scrolling through the different groups in the* ***Parameter*** *tree you can see the units selected for the parameters. Although most of the parameters have convenient units such as ft/s for velocity and CFM (cubic feet per minute) for volume flow rate we will change a couple of units to that are more convenient for this model. Since the physical size of the model may be relatively small it is more convenient to choose inches instead of feet as the length unit.*

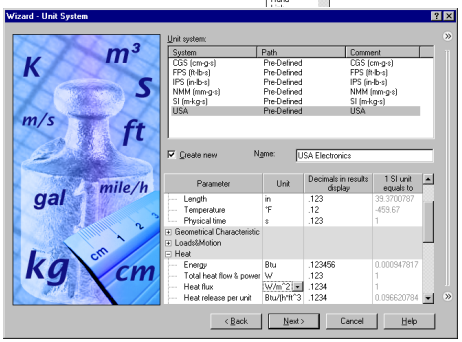
4 For the **Length** entry, double-click its cell in the **Unit** column and select **Inch**.



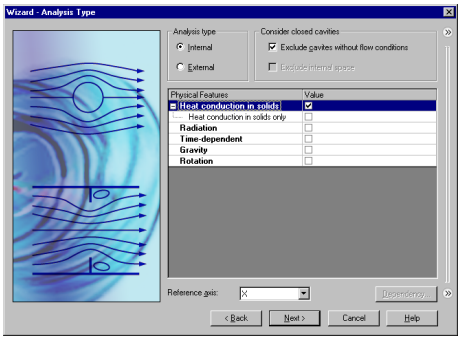
5 Next expand the **Heat** group in the **Parameter** tree.

Select **Watt**, **Watt/meter^2**, **Watt/meter^2/Kelvin** as the units for **Total heat flow and power**, **Heat flux** and **Heat transfer coefficient** respectively, because these units are more convenient when dealing with electronic components .

Click **Next**.



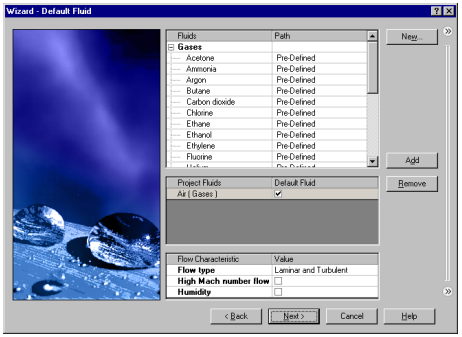
6 Set the analysis type to **Internal**. Under **Physical Features** select the **Heat conduction in solids** check box, then click **Next**.



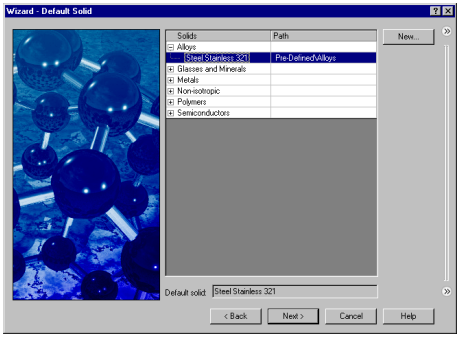
🕮 *Heat conduction in solids is selected because heat is generated by several electronics components and we are interested to see how the heat is dissipated through the heat sink and other solid parts and then out to the fluid.*

7 Expand the **Gases** folder and double-click **Air**. Keep the default **Flow Characteristics**.

Click **Next**.



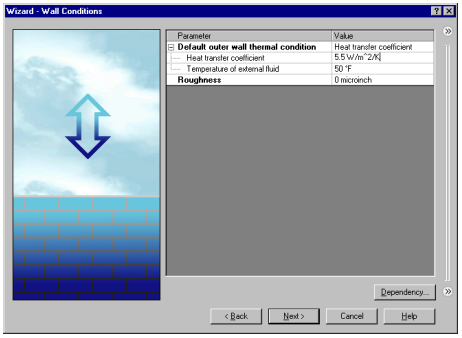
8 Expand the **Alloys** folder and click **Steel Stainless 321** to assign it as the **Default solid**.



🕮 *In the **Wizard** you specify the default solid material applied to all solid components in the FloEFD project. To specify a different solid material for one or more components, you can define a **Solid Material** condition for these components after the project is created.*

Click **Next**.

9 Select **Heat transfer coefficient** as **Default outer wall thermal condition** and specify the **Heat transfer coefficient** value of `5.5 W/m^2/K` and **Temperature of external fluid** of `50°F`. The entered value of heat transfer coefficient is automatically converted to the selected system of units (USA Electronics).

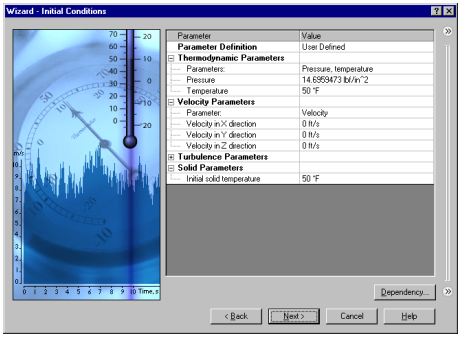


🕮 *In the **Wall Conditons** dialog box of the **Wizard** you specify the default conditions at the model walls. When **Heat conduction in solids** is enabled in an internal analysis, the **Default outer wall thermal condition** parameter allows you to simulate heat exchange between the outer model walls and surrounding environment. In our case the box is located in an air-conditioned room with the air temperature of* 50°F *and heat transfer through the outer walls of the enclosure due to the convection in the room can significantly contribute to the enclosure cooling.*

Click **Next**.

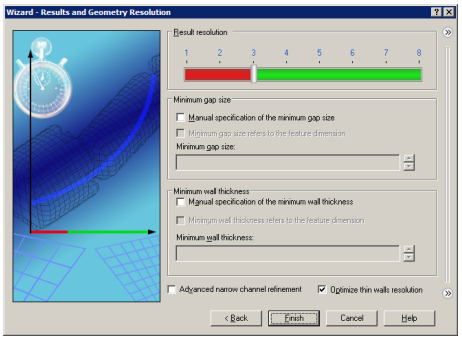
Although the initial temperature is more important for transient calculations to see how much time it takes to reach a certain temperature, in a steady-state analysis it is useful to set the initial temperature close to the expected final solution to speed up convergence. In this case we will set the initial air temperature and the initial temperature of the stainless steel (which represents the material of enclosure) to 50°F because the box is located in an air-conditioned room.

10 Set the initial fluid **Temperature** and the **Initial solid temperature** (under **Solid Parameters**) to `50°F`.



Click **Next**.

11 Accept the default **Result resolution** and keep the automatic evaluation of the **Minimum gap size** and **Minimum wall thickness**.



📖 *FloEFD calculates the default minimum gap size and minimum wall thickness using information about the overall model dimensions, the computational domain, and dimensions of faces on which you specify conditions and goals. Prior to starting the calculation, we recommend you to check the minimum gap size and minimum wall thickness to ensure that small features will be recognized. We will review this again after all the necessary conditions and goals are specified.*

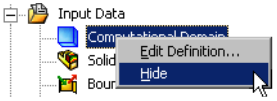
Click **Finish**. Now FloEFD creates a new project with the FloEFD data attached.

Switch to the FloEFD Analysis tab by clicking **FloEFD** .

Click to hide the FloEFD projects tree.

📖 *We will use the FloEFD Analysis tree to define our analysis, just as you use the Model Tree to design your models.*

Right-click the **Computational Domain** icon and select **Hide** to hide the wireframe box.



Specifying the Fan

A Fan is one of the types of boundary condition that defines the flow. You can specify **Fans** on the surfaces, free of **Boundary Conditions** and **Heat Sources**. At model openings closed by lids you can specify Inlet or Outlet Fans. You can also specify fans on any faces within the flow region as Internal Fans. A Fan is considered as an ideal device creating a flow with a certain volume (or mass) flow rate, which depends on the difference between the inlet and outlet pressures on the selected faces.

If you analyze a model with a fan, you should be familiar with the fan characteristics. In this example, we use one of the pre-defined fans available in the **Engineering Database**. If you cannot find an appropriate fan in the Engineering Database, you can define your own fan in accordance with the fan specifications.

1 Click **Flow Analysis > Insert > Fan**. The **Fan** dialog box appears.

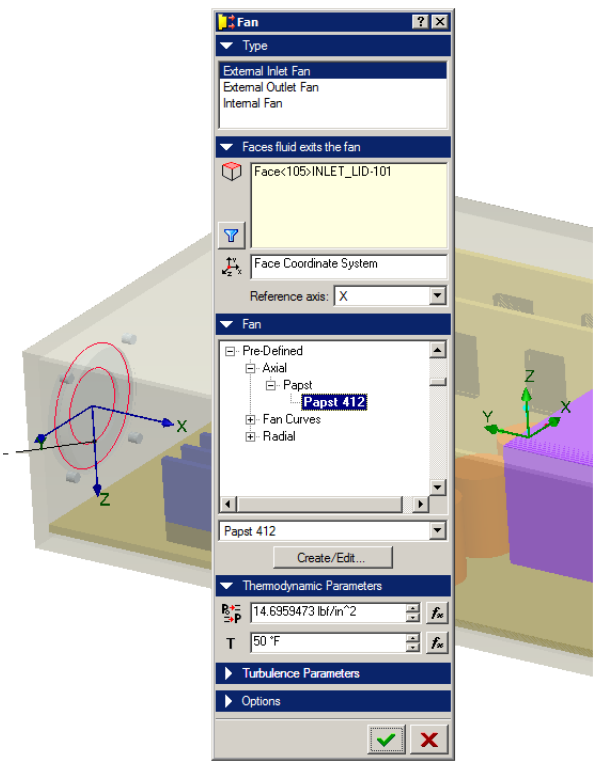
2 Select the inner face of the INLET_LID part as shown. (To access the inner face, set Filter to **Geometry** and right-click the **INLET_LID** until the inner face is highlighted, then click the left mouse button).

3 Under **Type**, select **External Inlet Fan**.

4 In the **Fan** list, under **Pre-Defined / Axial / Papst**, select the **Papst 412** item.

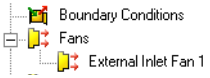
5 Under **Thermodynamic Parameters** check that the **Environment Pressure** corresponds to the atmospheric pressure.

6 Accept **Face Coordinate System** as the reference **Coordinate system** and **X** as the **Reference axis**.

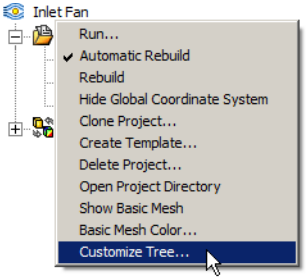


📖 ***Face coordinate system*** *is created automatically in the center of a planar face when you select this face as the face to apply the boundary condition or fan. The* ***X*** *axis of this coordinate system is normal to the face. The* ***Face coordinate system*** *is created only when one planar face is selected.*

7 Click **OK** ✔. The new **Fans** folder and the **External Inlet Fan 1** item appear in the FloEFD Analysis tree.



🕮 *Now you can edit the **External Inlet Fan 1** item or add a new fan using FloEFD Analysis tree. This folder remains visible until the last feature of this type is deleted. You can also make a feature folder to be initially available in the tree. Right-click the project name item and select **Customize Tree** to add or remove folders.*

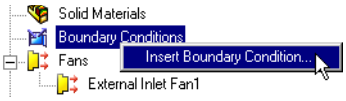


🕮 *Since the outlet lids of the enclosure are at ambient atmospheric pressure, the pressure rise produced by the fan is equal to the pressure drop through the electronics enclosure.*

Specifying Boundary Conditions

A boundary condition is required in any place where fluid enters or exits the model, excluding openings where a fan is specified. A boundary condition can be set in form of **Pressure, Mass Flow Rate**, **Volume Flow Rate** or **Velocity**. You can also use the **Boundary Condition** dialog for specifying an **Ideal Wall** condition that is an adiabatic, frictionless wall or a **Real Wall** condition to set the wall roughness and/or temperature and/or heat conduction coefficient at the selected model surfaces. For internal analyses with **Heat conduction in solids** enabled, you can also set thermal wall condition on outer model walls by specifying an **Outer Wall** condition.

1 In the FloEFD analysis tree right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.



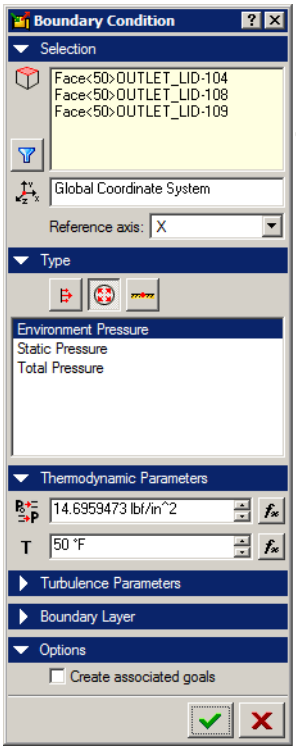
2 Select the inner faces of all outlet lids as shown.

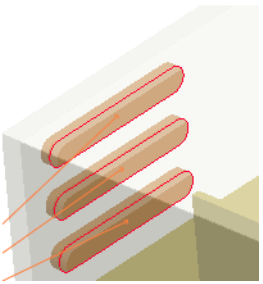
3 Select **Pressure Openings** and **Environment Pressure**.

📖 *The Environment pressure condition is interpreted as a static pressure for outgoing flows and as a total pressure for incoming flows.*

4 Keep the defaults under **Thermodynamic Parameters**, **Turbulence Parameters**, **Boundary Layer** and **Options**.

5 Click **OK**. The new **Environment Pressure 1** item appears in the FloEFD Analysis tree.





Specifying Heat Sources

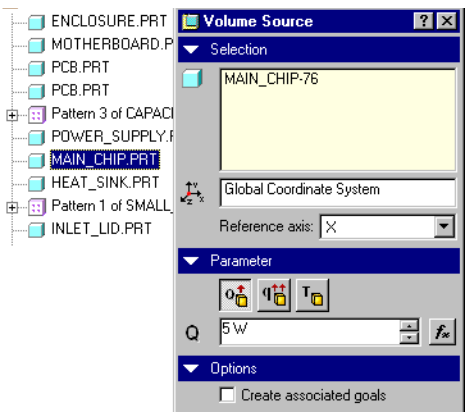
1 Click **Flow Analysis > Insert > Volume Source**.

2 Select the **MAIN_CHIP** from the Model tree to add it to the **Components to Apply the Volume Source** list.

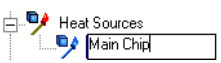
3 Select the **Heat Generation Rate** as **Parameter**.

4 Enter `5 W` in the **Heat Generation Rate** box.

5 Click **OK**.



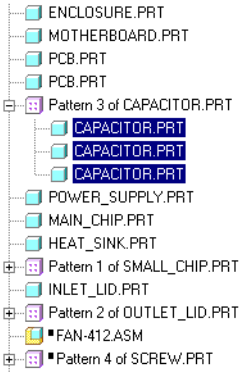
6 In the FloEFD Analysis tree, click-pause-click the new **VS Heat Generation Rate 1** item and rename it to `Main Chip`.



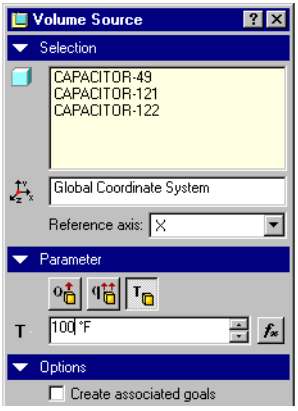
🕮 *Volume Heat Sources allow you to specify the heat generation rate (e.g. in Watts) or the volumetric heat generation rate (e.g. in Watts per volume) or a constant temperature boundary condition for a volume. It is also possible to specify Surface Heat Sources in terms of heat transfer rate (e.g. in Watts) or heat flux (e.g. in Watts per area).*

7 In the FloEFD analysis tree, right-click the **Heat Sources** icon and select **Insert Volume Source**.

8 In the Model tree, select all three **CAPACITOR** components under the **Pattern 3 of CAPACITOR** item.



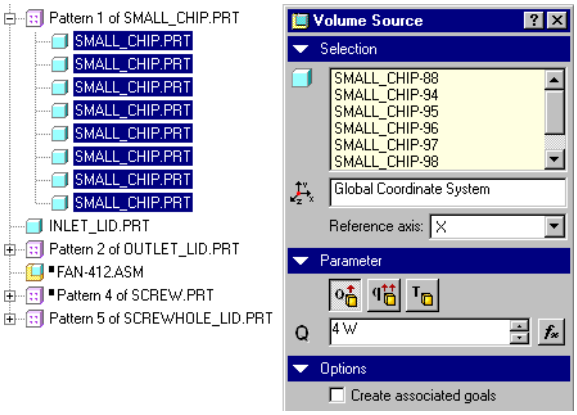
9 Select the **Temperature** as **Parameter** and enter 100 °F in the **Temperature** T box.

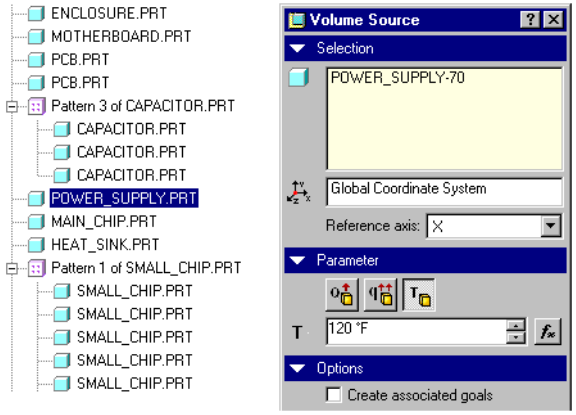


10 Click **OK** .

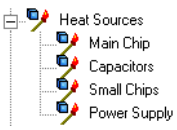
11 Click-pause-click the new **VS Temperature 1** item and rename it to `Capacitors`.

12 Following the same procedure as described above, specify the other volume heat sources as follows: all chips on PCBs (**SMALL_CHIP** components) with the total heat generation rate of 4 W, **POWER_SUPPLY** with the temperature of 120 °F.





13 Rename the heat source applied to the chips to `Small Chips` and the heat source for the power supply to `Power Supply`.



Click **File > Save**.

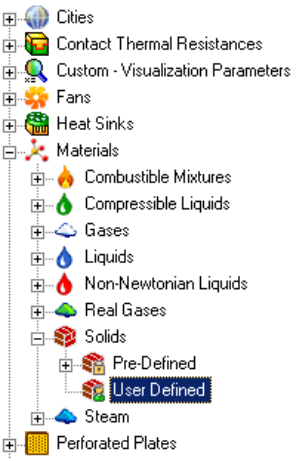
Creating Solid Materials in the Engineering Database

The real PCBs are made of laminate materials consisting of several layers of thin metal conductor interleaved with layers of epoxy resin dielectric. As for most laminate materials, the properties of a typical PCB material can vary greatly depending on the direction - along or across the layers, i.e. it is anisotropic. The Engineering Database contains some predefined PCB materials with anisotropic thermal conductivity.

In this tutorial example anisotropic thermal conductivity of PCBs does not affect the overall cooling performance much, so we will create a PCB material having the same thermal conductivity in all directions to learn how to add a new material to the Engineering Database and assign it to a part.

1 Click **Flow Analysis > Tools > Engineering Database**.

2 In the **Database tree** select **Materials / Solids / User Defined**.



3 Click **New Item** on the toolbar.

The blank **Item Properties** tab appears. Double-click the empty cells to set the corresponding properties values.

4 Specify the material properties as follows:

Name = `Tutorial PCB`,
Comments = `Isotropic PCB`,
Density = `1120 kg/m^3`,
Specific heat = `1400 J/(kg*K)`,
Conductivity type = `Isotropic`
Thermal conductivity = `10 W/(m*K)`,
Melting temperature = `390 K`.

Property	Value
Name	Tutorial PCB
Comments	Isotropic PCB
Density	1120 kg/m^3
Specific heat	1400 J/(kg*K)
Conductivity type	Isotropic
Thermal conductivity	10 W/(m*K)
Melting temperature	390 K

We also need to add a new material simulating thermal conductivity and other thermal properties of electronic components.

5 Switch to the **Items** tab and click **New Item** on the toolbar.

6 Specify the properties of the chips material:

Name = `Tutorial component package`,
Comments = `Component package`,
Density = `2000 kg/m^3`,
Specific heat = `120 J/(kg*K)`,
Conductivity type = `Isotropic`
Thermal conductivity = `0.4 W/(m*K)`,
Melting temperature = `390 K`.

Property	Value
Name	Tutorial component package
Comments	Component package
Density	2000 kg/m^3
Specific heat	120 J/(kg*K)
Conductivity type	Isotropic
Thermal conductivity	0.4 W/(m*K)
Melting temperature	390K

7 Click **Save**.

8 Click **File > Exit** to exit the database.

📖 *You can enter the material properties in any unit system you want by typing the unit name after the value and FloEFD will automatically convert the entered value to the SI system of units. You can also specify temperature-dependent material properties using the* **Tables and Curves** *tab.*

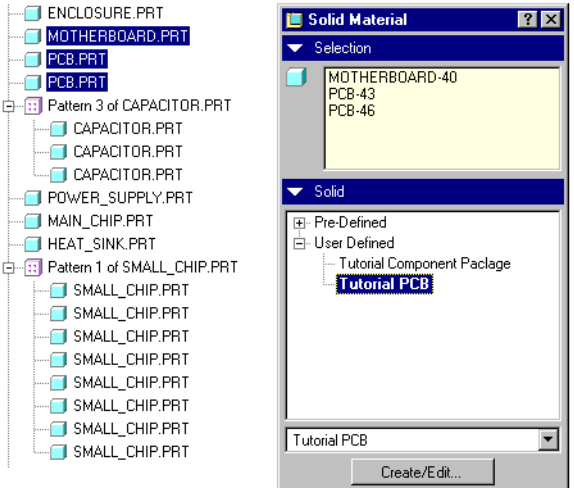
Specifying Solid Materials

The **Solid Material** feature is used to specify the material for solid parts in the assembly.

1 In the FloEFD analysis tree, right-click the **Solid Materials** icon and select **Insert Solid Material**.

2 In the Model Tree, select the **MOTHERBOARD** and both **PCB** components.

3 In the **Solid** list, expand **User Defined** and select **Tutorial PCB**.



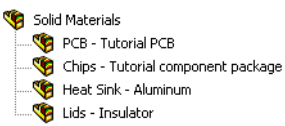
4 Click **OK** ✔.

5 Following the same procedure, specify solid materials for other components:

- for the **Main Chip** and all **Small Chips** assign the new **Tutorial component package** material (available under **User Defined**);
- the **Heat Sink** is made of **Aluminum** (available under **Pre-Defined / Metals**);
- the lids (**INLET_LID** and all lids in both the **Pattern 2 of OUTLET_LID** and **Pattern 5 of SCREWHOLE_LID** patterns) are made of the **Insulator** material (available under **Pre-Defined / Glasses and Minerals**).

To select a part, click it in the Model Tree or Creo Elements/Pro & Creo Parametric graphics area.

6 Change the name of each assigned solid material. The new, descriptive names should be:
`PCB - Tutorial PCB,`
`Chips - Tutorial component package,`
`Heat Sink - Aluminum,`
`Lids - Insulator.`

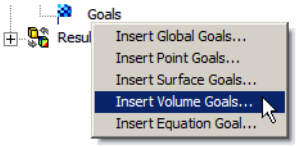


7 Click **File > Save**.

Specifying Engineering Goals

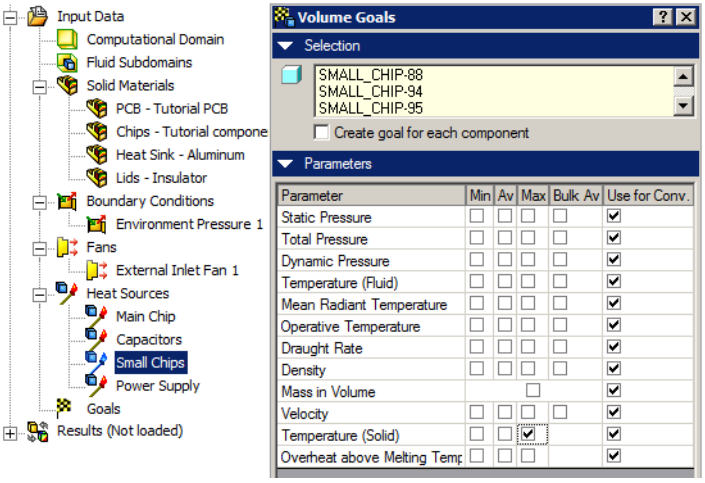
Specifying Volume Goals

8 In the FloEFD analysis tree, right-click the **Goals** icon and select **Insert Volume Goals**.



9 Select the **Small Chips** item in the FloEFD analysis tree. This will select all components belonging to the **Small Chips** heat source.

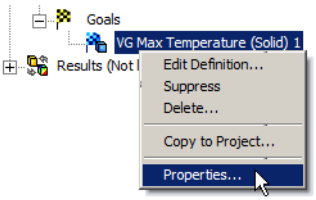
10 In the **Parameter** table, select the **Max** check box in the **Temperature (Solid)** row.



11 Accept selected **Use for Conv.** check box to use this goal for convergence control.

12 Click **OK**. The new **VG Max Temperature (Solid) 1** item appears in the FloEFD Analysis tree.

13 Change the name of the new item to `VG Small Chips Max Temperature`. You can also change the name of the item using the **Feature Properties** dialog that appears if you right-click the item and select **Properties**.

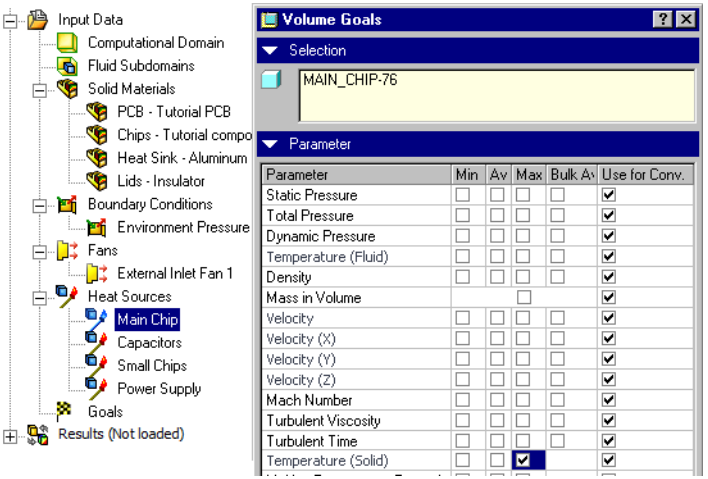


14 Right-click the **Goals** icon and select **Insert Volume Goals**.

15 Select the **Main Chip** item in the FloEFD analysis tree.

16 In the **Parameter** table, select the **Max** check box in the **Temperature (Solid)** row.

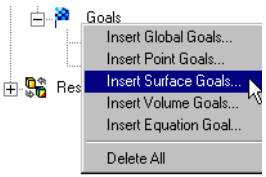
17 Click **OK**.



18 Rename the new **VG Max Temperature (Solid) 1** item to `VG Chip Max Temperature`.
Click anywhere in the graphic area to clear the selection.

Specifying Surface Goals

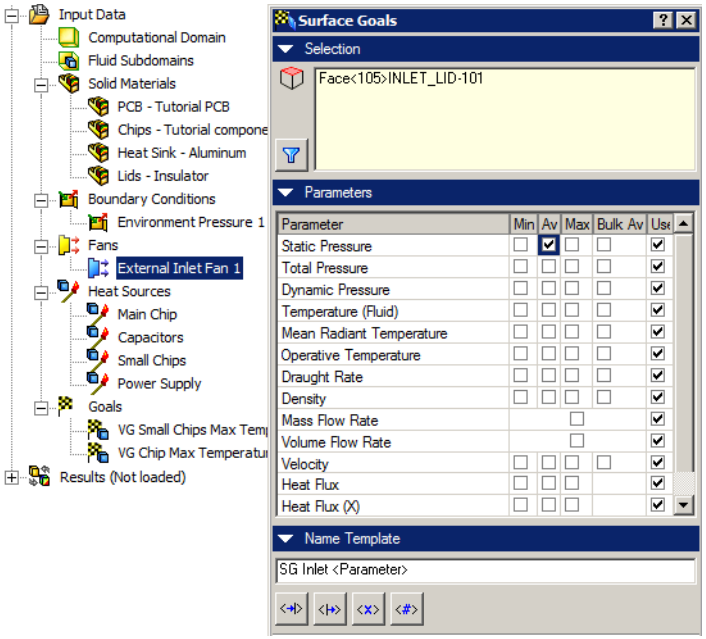
1 Right-click the **Goals** icon and select **Insert Surface Goals**.



2 Click the **External Inlet Fan 1** item to select the face where the goal is going to be applied.

3 In the **Parameter** table select the **Av** check box in the **Static Pressure** row.

4 Accept selected **Use for Conv.** check box to use this goal for convergence control.



📖 *Notice the X(Y, Z) - Component of Force and X(Y, Z) - Component of Torque surface goals. For these you can select the Coordinate system, in which these goals will be calculated.*

5 At the bottom of the **Surface Goals** dialog box click **Inlet** and then remove the `<Number>` field from the **Name Template** box.

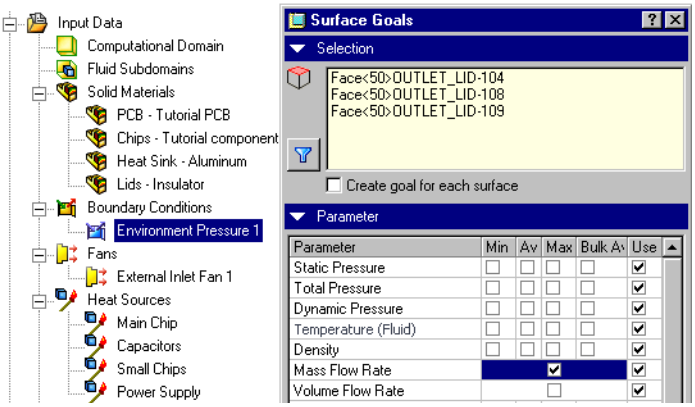
6 Click **OK**.

The new **SG Inlet Av Static Pressure** goal appears.
Click anywhere in the graphic area to clear the selection.

1 Right-click the **Goals** icon and select **Insert Surface Goals**.

2 Click the **Environment Pressure 1** item to select the faces where the goal is going to be applied.

3 In the **Parameter** table select the first check box in the **Mass Flow Rate** row.



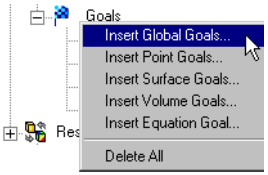
4 Accept selected **Use for Conv.** check box to use this goal for convergence control.

5 At the bottom of the **Surface Goals** dialog box, click **Outlet** and then remove the `<Number>` field from the **Name Template**.

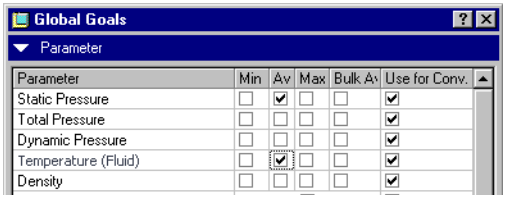
6 Click **OK**. The **SG Outlet Mass Flow Rate** goal appears.

Specifying Global Goals

1 Right-click the **Goals** icon and select **Insert Global Goals**.



2 In the **Parameter** table select the **Av** check boxes in the **Static Pressure** and **Temperature (Fluid)** rows and accept selected **Use for Conv.** check box to use these goals for convergence control.



3 Remove the `<Number>` field from the **Name Template** and click **OK**. The **GG Av Static Pressure** and **GG Av Temperature (Fluid)** goals appear.In this tutorial, the engineering goals are set to determine the maximum temperature of the heat generating components, the temperature rise in air and the pressure drop and mass flow rate through the enclosure.

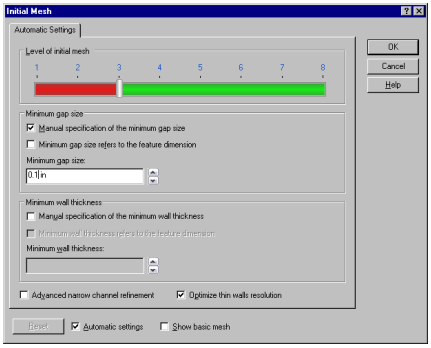
Goals
GG Av Static Pressure
GG Av Temperature (Fluid)
SG Inlet Av Static Pressure
SG Outlet Mass Flow Rate
VG Chip Max Temperature
VG Small Chips Max Temperature

4 Click **File > Save**.

Next let us check the automatically defined geometry resolution settings for this project.

Changing the Geometry Resolution

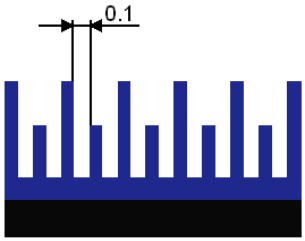
1 Click **Flow Analysis > Initial Mesh**.



*In Creo Parametric click **Flow Analysis > Mesh > Initial Mesh**.*

2 Select the **Manual specification of the minimum gap size** check box.

3 Enter `0.1 in` for the **Minimum gap size** (i.e. passage between the fins of the heat sink).



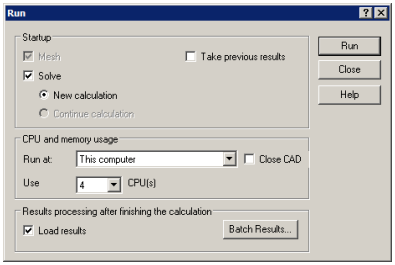
*Entering values for the minimum gap size and minimum wall thickness is important when you have small features. Setting these values accurately ensures that the small features are not "passed over" by the mesh. The minimum wall thickness should be specified only if there are fluid cells on either side of a small solid feature. In case of internal analyses, there are no fluid cells in the ambient space outside of the enclosure. Therefore boundaries between internal flow and ambient space are always resolved properly. That is why you should not take into account the walls of the steel cabinet. Both the **minimum gap size** and the **minimum wall thickness** are tools that help you to create a model-adaptive mesh resulting in increased accuracy. However the minimum gap size setting is the more powerful one. The fact is that the FloEFD mesh is constructed so that the specified Level of initial mesh controls the minimum number of mesh cells per **minimum gap size**. And this number is equal to or greater than the number of mesh cells generated per **minimum wall thickness**. That's why even if you have a thin solid feature inside the flow region it is not necessary to specify minimum wall thickness if it is greater than or equal to the minimum gap size. Specifying the minimum wall thickness is necessary if you want to resolve thin walls smaller than the smallest gap.*

4 Click **OK**.

Running the Calculation

1 Click **Flow Analysis > Solve > Run**.

2 Click **Run**.



It will take a few minutes to calculate this problem on a typical PC.

🕮 *You may notice that different goals take different number of iterations to converge. The goal-oriented philosophy of FloEFD allows you to get the answers you need in the shortest amount of time. For example, if you were only interested in the temperature of fluid in the enclosure, FloEFD would have provided the result more quickly then if the solver was allowed to fully converge on all of the parameters.*

Viewing the Goals

1 Right-click the **Goal Plots** icon under **Results** and select **Insert**.

2 In the **Goals** dialog, select **All**.

3 Click **OK**.

An Excel spreadsheet with the goal results will be open. The first sheet will show a table summarizing the goals.

ENCLOSURE_ASSEMBLY.ASM [INLET_FAN]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value
GG Av Static Pressure	[lbf/in^2]	14,69677107	14,69677231	14,69677092	14,69677308
SG Inlet Av Static Pressure	[lbf/in^2]	14,69653289	14,69653726	14,69653289	14,69653939
GG Av Temperature (Fluid)	[°F]	61,37009916	61,30920925	61,18051182	61,40785695
SG Outlet Mass Flow Rate	[lb/s]	-0,007193183	-0,007190625	-0,007193183	-0,007189949
VG Small Chips Max Temp	[°F]	89,24271231	89,14410831	88,65386757	89,445698
VG Chip Max Temperature	[°F]	88,33986905	88,34641545	88,31882243	88,36898683

You can see that the maximum temperature in the main chip is about 88 °F, and the maximum temperature over the small chips is about 89 °F.

🕮 *Goal progress bar is a qualitative and quantitative characteristic of the goal convergence process. When FloEFD analyzes the goal convergence, it calculates the goal's amplitude excursion of the averaged value defined as the difference between the maximum and minimum values of the averaged goal over the analysis interval reckoned from the last iteration and compares this amplitude excursion with the goal's*

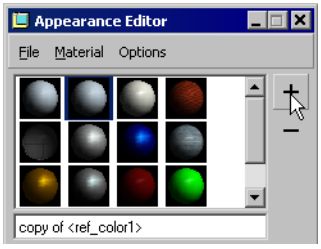
convergence criterion amplitude excursion, either specified by you or automatically determined by [Product] as a fraction of the goal's physical parameter amplitude excursion over the analysis interval reckoned from the fourth iteration until one travel is completed. The percentage of the goal's convergence criterion to the goal's real amplitude excursion over the analysis interval is shown in the goal's convergence progress bar (when the goal's real amplitude excursion becomes equal or smaller than the goal's convergence criterion, the progress bar is replaced by word "Achieved"). Naturally, if the goal's real amplitude excursion oscillates, the progress bar oscillates also, moreover, when a hard problem is solved, it can noticeably regress, in particular from the "achieved" level. The calculation can finish if the iterations (in travels) required for finishing the calculation have been performed, or if the goal convergence criteria are satisfied before performing the required number of iterations. You can specify other finishing conditions at your discretion.

To analyze the results in more detail let us use the various FloEFD results processing tools. The best method for the visualization of how the fluid flows inside the enclosure is to create flow trajectories.

Adjusting Model Transparency

Click **View > Color and Appearance**.

In the **Appearance Editor** click the plus button to add a new appearance.



Go to **Advanced** tab and adjust transparency to about 80.

Click **Apply** and then **Close**.

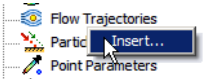
📖 *The first step for results processing is to create a transparent view of the geometry, a 'glass-body'. This way you can easily see where cut planes etc. are located with respect to the geometry.*

✎ *In Creo Parametric click **View > Model Display > Appearance Galery > More Appearances**. In the **Appearance Editor** adjust transparency to about 80.*

Viewing Flow Trajectories

1 Right-click the **Flow Trajectories** icon and select **Insert**.

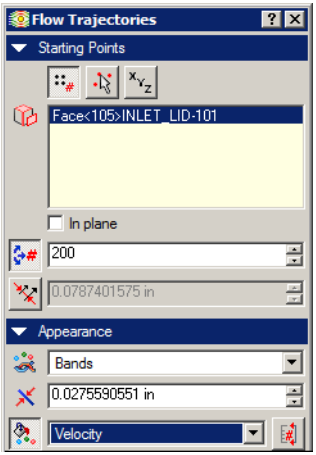
2 In the FloEFD Analysis click the **External Inlet Fan1** item to select the inner face of the **INLET_LID**.



3 Set the **Number of Points** to 200.

4 Under **Appearance**, set **Draw Trajectories as Bands**.

5 Make sure that **Color by Parameter** is selected and then change the parameter to **Velocity**.

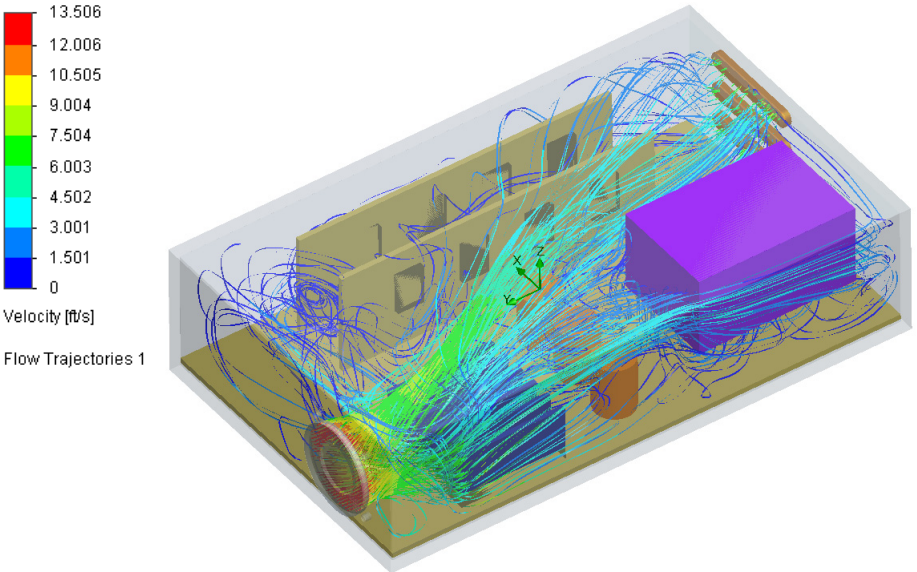


If **Color by Parameter** *is selected, then the trajectories are colored in accordance with the distribution of the parameter specified. If you select* **Color** *then all flow trajectories will have a fixed color specified by you.*

6 Click **OK**.

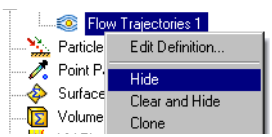
The new **Flow Trajectories 1** item appears in the FloEFD Analysis tree.

This is the picture you should see.



Notice that there are only a few trajectories along the adjacent to the wall **PCB** and this may cause problems with cooling of the chips placed on this PCB. Additionally the blue color indicates low velocity in front of this **PCB** .

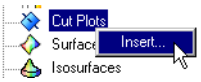
Right-click the **Flow Trajectories 1** item and select **Hide**.



Let us now examine the velocity distribution in more detail.

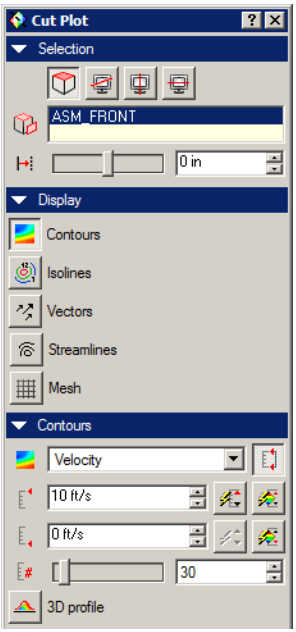
Viewing Cut Plots

1 Right-click the **Cut Plots** icon and select **Insert**.



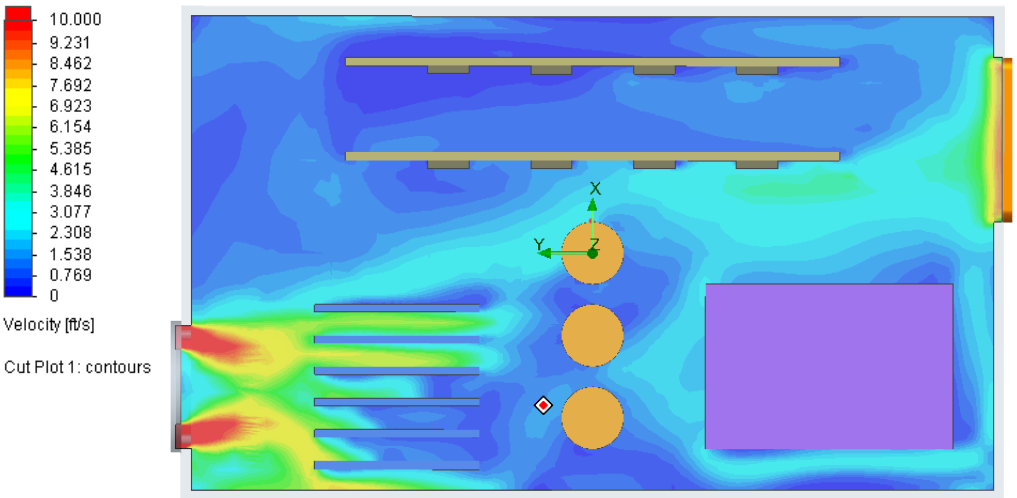
2 Select the ASM_FRONT plane as the section plane.

3 Under **Contours**, change the parameter to **Velocity**, then select **Adjust Minimum and Maximum**. Change the **Min** and **Max** values to `0` and `10 ft/s` respectively. The specified values produce a palette where it is easier to determine the value.



4 Set the **Number of levels** to `30`.

5 Click **OK**. The new **Cut Plot 1** item appears in the FloEFD Analysis tree.



Let us now look at the fluid temperature.

6 Right-click the **Cut Plot 1** icon and select **Edit Definition**.

7 Change the **Offset** to `-0.3 in`.

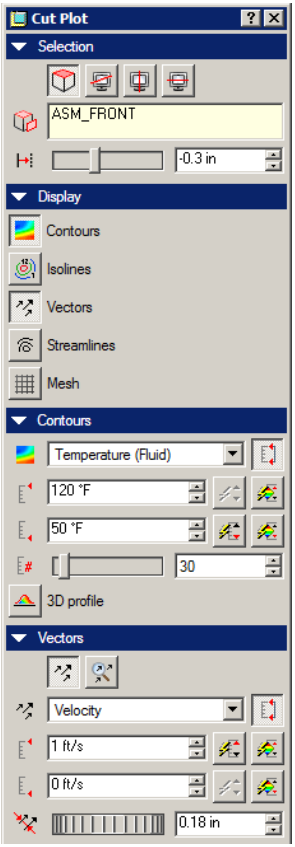
8 Change the **Parameter** from **Velocity** to **Temperature (Fluid)**.

9 Change the **Min** and **Max** values to `50` and `120 F` respectively.

10 Under **Display**, select **Vectors**.

11 Under the appeared **Vectors** tab, make sure that **Static Vectors** is selected and the **Parameter** is set to **Velocity**, then select **Adjust Minimum and Maximum**.

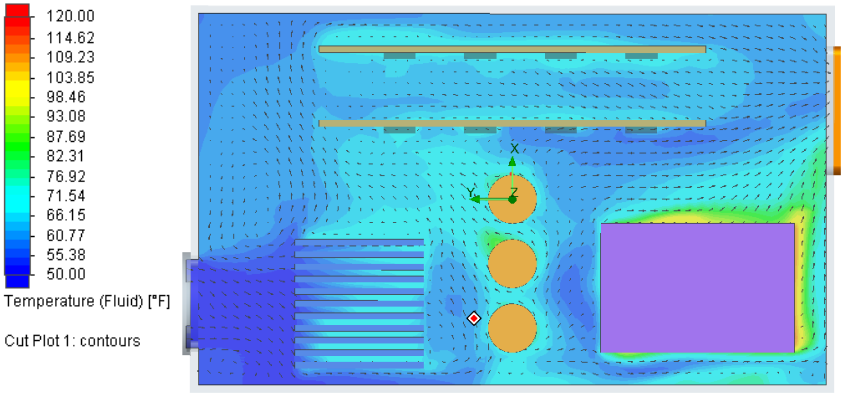
12 Set the **Max** value to `1 ft/s`.



📖 *By specifying the custom **Min** and **Max** values you can control the vector length. The vectors whose velocity exceeds the specified Max value will have the same length as the vectors whose velocity is equal to Max. Likewise, the vectors whose velocity is less than the specified Min value will have the same length as the vectors whose velocity is equal to Min. We have set 1 ft/s in order to display areas of low velocity.*

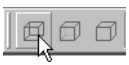
13 Change the **Spacing** to 0.18 in.

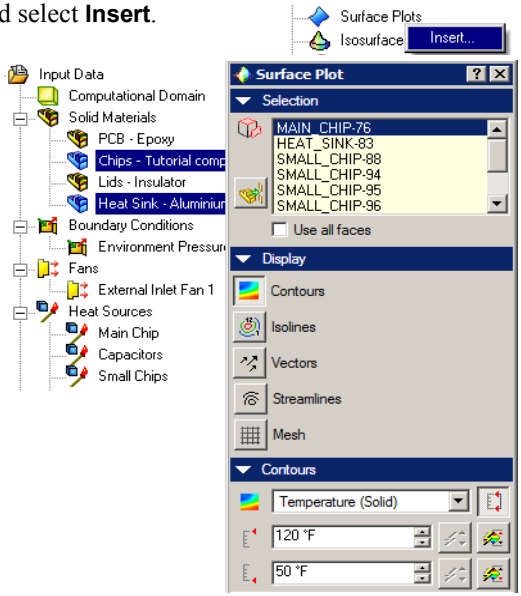
14 Click **OK**.

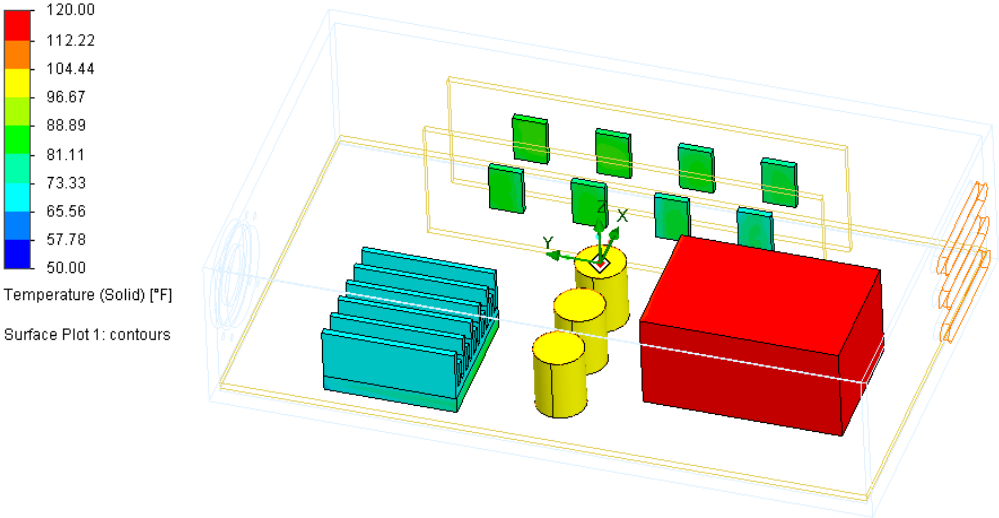


Right-click the **Cut Plot 1** item and select **Hide**. Let us now display solid temperature.

Viewing Surface Plots

1. Right-click the **Surface Plots** item and select **Insert**.
2. Hold down the **Ctrl** key and select **Heat Sink - Aluminum** and **Chips - Tutorial component package** items in the Analysis Tree.
3. Under **Contours**, change the **Parameter** to **Temperature (Solid)**.
4. Change the **Min** and **Max** values to `50` and `120 F` respectively.
5. Click **OK**.
6. Repeat steps 1 and 2 and select the **Power Supply** and **Capacitors** items, then click **OK**.
7. On the **Model Display** toolbar click **Wireframe**  to show only the face outlines.





You can view and analyze the results further with the post-processing tools that were shown in the *Ball Valve Design* tutorial. FloEFD allows you to quickly and easily investigate your design both quantitatively and qualitatively. Quantitative results such as the maximum temperature in the component, pressure drop through the cabinet, and air temperature rise will allow you to determine whether the design is acceptable or not. By viewing qualitative results such as air flow patterns, and heat conduction patterns in the solid, FloEFD gives you the necessary insight to locate problem areas or weaknesses in your design and provides guidance on how to improve or optimize the design.

A3

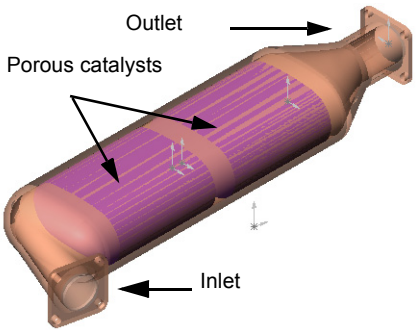
Porous Media

In this tutorial we consider flow in a section of an automobile exhaust pipe, whose exhaust flow is resisted by two porous bodies serving as catalysts for transforming harmful carbon monoxide into carbon dioxide. When designing an automobile catalytic converter, the engineer faces a compromise between minimizing the catalyst's resistance to the exhaust flow while maximizing the catalyst's internal surface area and duration that the exhaust gases are in contact with that surface area. Therefore, a more uniform distribution of the exhaust mass flow rate over the catalyst's cross sections favors its serviceability. The porous media capabilities of FloEFD are used to simulate each catalyst, which allows you to model the volume that the catalyst occupies as a distributed resistance instead of discretely modeling all of the individual passages within the catalyst, which would be impractical or even impossible. Here, as a FloEFD tutorial example we consider the influence of the catalysts' porous medium permeability type (isotropic and unidirectional media of the same resistance to flow) on the exhaust mass flow rate distribution over the catalysts' cross sections. We will observe the latter through the behavior of the exhaust gas flow trajectories distributed uniformly over the model's inlet and passing through the porous catalysts. Additionally, by coloring the flow trajectories by the flow velocity the exhaust gas residence time in the porous catalysts can be estimated, which is also important from the catalyst effectiveness viewpoint.

Opening the Model

1 Copy the **A3 - Porous Media** folder into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files. Run FloEFD. Click **File > Open.**

2 In the **File Open** dialog box, browse to the `catalyst.asm` assembly located in the **A3 - Porous Media** folder and click **Open** (or double-click the assembly).

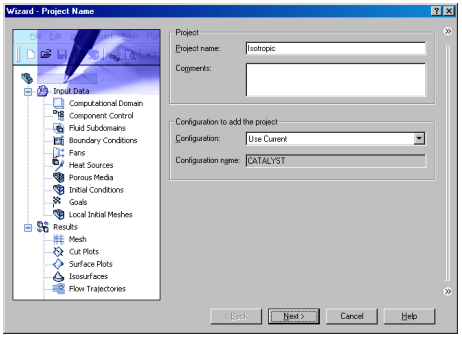


📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the* `catalyst.asm` *assembly located in the* ***A3 - Porous Media\Ready To Run*** *folder and run the desired projects.*

Creating a FloEFD Project

1 Click **Flow Analysis > Project > Wizard**.

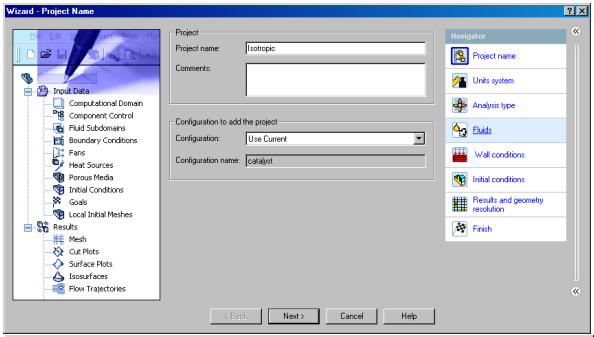
Once inside the **Wizard**, type a project name: `Isotropic`. Under **Configuration to add the project**, keep **Use Current**.



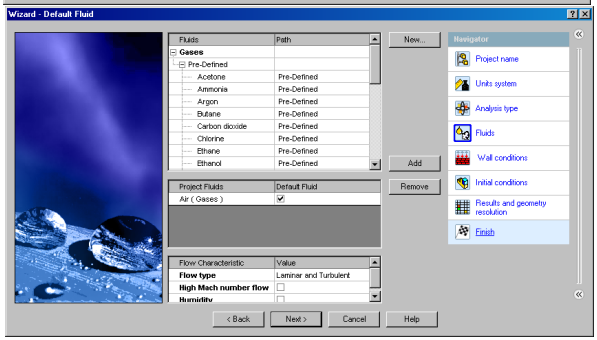
📖 *The project Wizard guides you through the definition of the project's properties step-by-step. Except for two steps (steps to define the project fluids and default solid), each step has some pre-defined values, so you can either accept these values (skipping the step by clicking* **Next***) or modify them to your needs.*
These pre-defined settings are:
unit system – SI,
analysis type – internal, no additional physical capabilities are considered,
wall condition – adiabatic wall
initial conditions – pressure - 1 atm, temperature - 293.2 K.
result and geometry resolution – level 3,
For this project these default settings suit perfectly and all what we need to do is just to select Air as the project fluid. To avoid passing through all steps we will use ***Navigator*** *pane that provides a quick access to the Wizard's pages.*

2 Click an arrow » at the right.

3 In the **Navigator** pane click **Fluids**.



4 Open the **Gases** folder, click **Air**, then click **Add**.



5 Since we do not need to change other properties we can close the Wizard. Click **Finish** in the Navigator panel.

📖 *You can click Finish at any moment, but if you attempt to close Wizard without specifying all obligatory properties (such as project fluids), the Wizard will not close and the page where you need to define a missing property will be marked by the exclamation icon* ⚠.

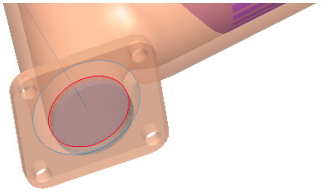
Now FloEFD creates a new project with the FloEFD data attached.

In the FloEFD Analysis tree, right-click the **Computational Domain** icon and select **Hide** to hide the black wireframe box.

Specifying Boundary Conditions

1 In the FloEFD Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

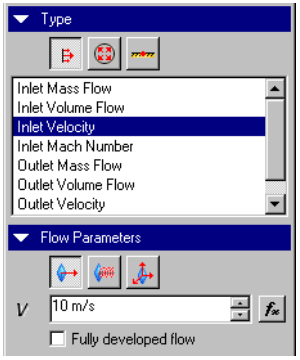
2 Select the inner face of the inlet lid as shown.



3 Select **Flow Openings** and **Inlet Velocity**.

4 Set the **Velocity Normal to Face** to `10 m/s`.

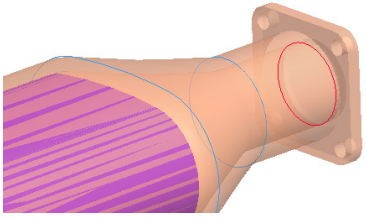
5 Click **OK**.



📖 *With the definition just made, we told FloEFD that at this opening air is flowing into the catalyst with a velocity of 10 m/s.*

6 Select the inner face of the outlet lid as shown.

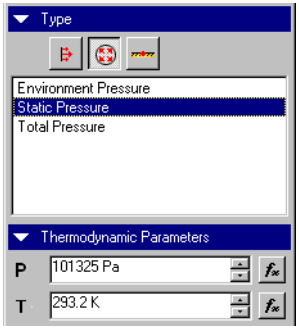
7 Right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.



8 Select **Pressure Openings** and **Static Pressure**.

9 Keep the defaults under **Thermodynamic Parameters**, **Turbulence Parameters**, **Boundary Layer** and **Options**.

10 Click **OK**.



📖 *With the definition just made, we told FloEFD that at this opening the fluid exits the model to an area of static atmospheric pressure.*

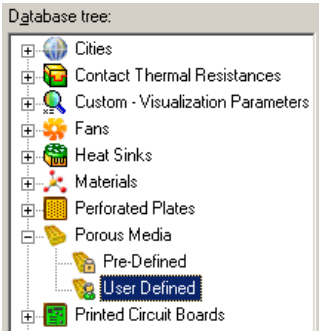
Now we can specify porous media in this project. To do this, first we need to specify the porous medium's properties (porosity, permeability type, etc.) in the **Engineering Database** and then apply this feature to the components in the assembly.

Creating Isotropic Porous Medium in the Engineering Database

The material you are going to create is already defined in the Engineering Database under the Pre-Defined folder. You can skip the definition of porous material and select the pre-defined "Isotropic" material from the Engineering database when you will assign the porous material to a component later in this tutorial.

1 Click **Flow Analysis > Tools > Engineering Database**.

2 In the **Database tree** select **Porous Media / User Defined**.



3 Click **New Item** on the toolbar. The blank **Item Properties** tab appears. Double-click the empty cells to set the corresponding property values.

4 Name the new porous medium `Isotropic`.

5 Under **Comment**, click the ... button and type the desired comments for this porous medium. The **Comment** property is optional, you can leave this field blank.

6 Set the medium's **Porosity** to `0.5`.

🕮 *Porosity is the effective porosity of the porous medium, defined as the volume fraction of the interconnected pores with respect to the total porous medium volume; here, the porosity is equal to 0.5. The porosity will govern the exhaust flow velocity in the porous medium channels, which, in turn, governs the exhaust gas residence in the porous catalyst and, therefore, the catalyst efficiency.*

7 Choose **Isotropic** as the **Permeability type**.

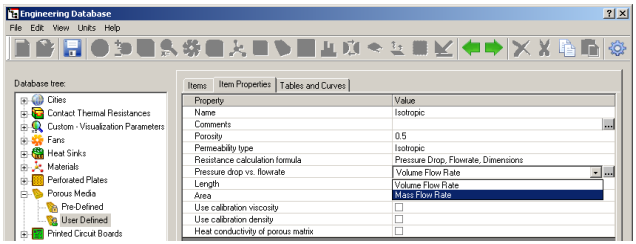
🕮 *First of all let us consider an **Isotropic** permeability, i.e, a medium with permeability not depending on the direction within the medium. Then, as an alternative, we will consider a **Unidirectional** permeability, i.e., the medium permeable in one direction only.*

8 Choose **Pressure drop**, **Flowrate**, **Dimensions** as the **Resistance calculation formula.**

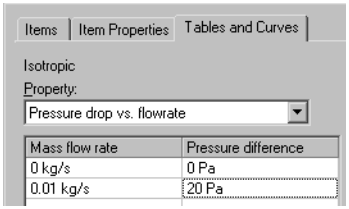
🕮 *For our media we select the* **Pressure Drop, Flowrate, Dimensions** *medium resistance to flow, i.e., specify the porous medium resistance as $k = \Delta P \times S / (m \times L)$ (in units of s^{-1}), where the right-side parameters are referred to a tested parallelepiped sample of the porous medium, having the cross-sectional area S and the length L in the selected sample direction, in which the mass flow rate through the sample is equal to m under the pressure difference of ΔP between the sample opposite sides in this direction. In this project we will specify $\Delta P = 20$ Pa at $m = 0.01$ kg/s (and $\Delta P = 0$ Pa*

at m=0 kg/s), S = 0.01 m^2, L = 0.1m. Therefore, k = 200 s^{-1}. Knowing S and L of the catalyst inserted into the model and m of the flow through it, you can approximately estimate the pressure loss at the model catalyst from $\Delta P = k \times m \times L/S$.

9 For the **Pressure drop vs. flowrate** choose **Mass Flow Rate**. Click the ... button to switch to the **Tables and Curves** tab.



10 In the **Property** table specify the linear dependency of pressure drop vs. mass flow rate as shown in the picture right.



11 Go back to the **Item Properties** tab.

12 Set **Length** to `0.1 m` and **Area** to `0.01 m`^{2}.

Pressure drop vs. flowrate	Mass Flow Rate
Length	0.1 m
Area	0.01 m^2

13 Click **Save** .

14 Click **File**, **Exit** to exit the database.

Now we will apply the specified porous medium to the model components representing the porous bodies.

Note that a porous medium is applied only to a component that is not treated by FloEFD as a solid body. By default, all the components in the assembly considered are treated as solids. If there is a component that is not supposed to be treated as solid, you have to disable it in the **Component Control** dialog box. Components are automatically disabled when you assign a porous media to them by creating the **Porous Medium** condition, so you do not need to disable them manually.

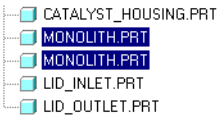
Specifying Porous Medium

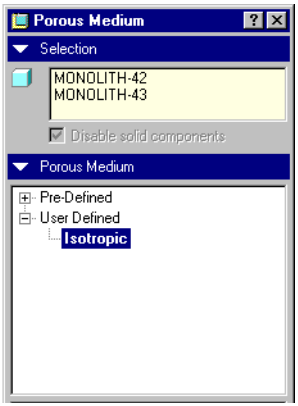
1 Click **Flow Analysis > Insert > Porous Medium**.

2 In the Model Tree, select both of the **MONOLITH** components.

3 Expand the list of the **User Defined** porous media and select **Isotropic**. If you skipped the definition of porous medium, use the **Isotropic** material available under **Pre-Defined**.

4 Click **OK** .

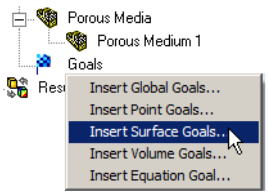




To obtain the total pressure drop between the model inlet and outlet we will specify an **Equation Goal** based on two **Surface Goals**.

Specifying Surface Goals

1 Right-click the **Goals** icon and select **Insert Surface Goals**.

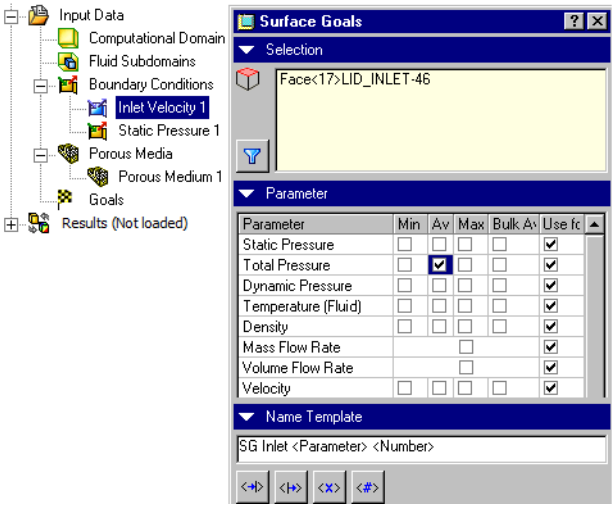


2 In the FloEFD Analysis tree, click the **Inlet Velocity 1** item to select the inner face of the inlet lid.

3 In the **Parameter** table select the **Av** check box in the **Total Pressure** row.

4 Accept the selected **Use for Conv.** check box to use this goal for convergence control.

5 At the bottom of the **Surface Goals** dialog box click **Inlet** .



6 Click **OK** - the new `SG Inlet Av Total Pressure 1` goal appears.

7 Right-click the **Goals** icon and select **Insert Surface Goals**.

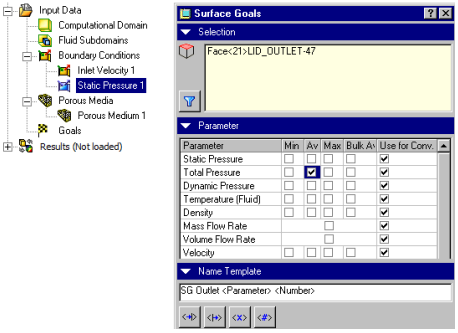
8 Click the **Static Pressure 1** item to select the inner face of the outlet lid.

9 In the **Parameter** table select the **Av** check box in the **Total Pressure** row.

10 Accept the selected **Use for Conv.** check box to use this goal for convergence control.

11 At the bottom of the **Surface Goals** dialog box click **Outlet** .

12 Click **OK** - the new `SG Outlet Av Total Pressure 1` goal appears.



Specifying the Equation Goal

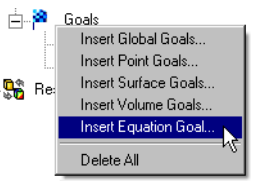
Equation Goal is a goal defined by an analytical function of the existing goals and/or parameters of input data conditions. This goal can be viewed as equation goal during the calculation and while displaying results in the same way as the other goals. As variables, you can use any of the specified goals, including another equation goals, except for goals that are dependent on other equation goals, and parameters of the specified project's input data features (general initial or ambient conditions, boundary conditions, fans, heat sources, local initial conditions). You can also use constants in the definition of the equation goal.

1 Right-click the **Goals** icon and select **Insert Equation Goal.**

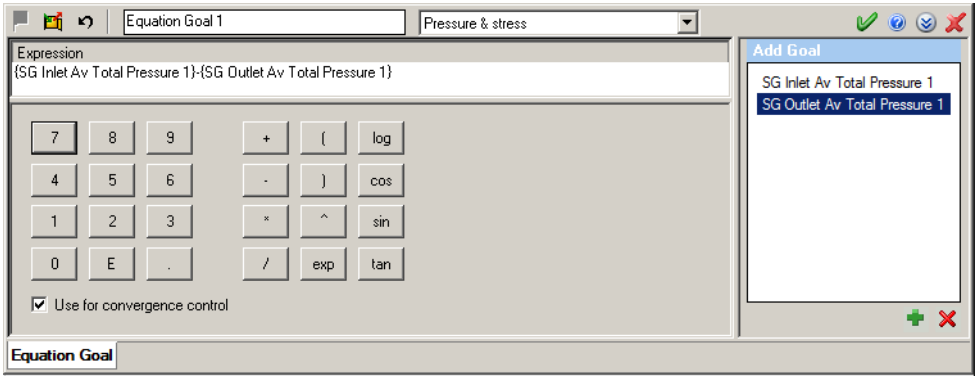
2 On the pane in the bottom of the screen click **Add Goal** .

3 From the **Add Goal** list select the **SG Inlet Av Total Pressure 1** goal and click **Add** . It appears in the **Expression** box.

4 Click the minus "-" button in the calculator.



5 From the **Add Goal** list select the **SG Outlet Av Total Pressure 1** goal and click **Add** . In the **Expression** box the resulting expression appears.


Equation Goal 1
Pressure & stress
Expression
{SG Inlet Av Total Pressure 1}-{SG Outlet Av Total Pressure 1}
Add Goal
SG Inlet Av Total Pressure 1
SG Outlet Av Total Pressure 1
7 8 9 + (log
4 5 6 -) cos
1 2 3 * ^ sin
0 E . / exp tan
Use for convergence control
Equation Goal

You can use goals (including previously specified Equation Goals), parameters of input data conditions and constants in the expression defining an Equation Goal. If the constants in the expression represent some physical parameters (i.e. length, area, etc.), make sure that they are specified in the project's system of units. FloEFD has no information about the physical meaning of the constants you use, so you need to specify the Equation Goal dimensionality by yourself.

To add an area or a volume of the model items (faces, components, etc.) as a variable, previously create a corresponding goal on the desired surfaces or components by using one of the following parameters: Area (Fluid), Area (Solid), Volume (Fluid), Volume (Solid) and then add the created goal as a variable.

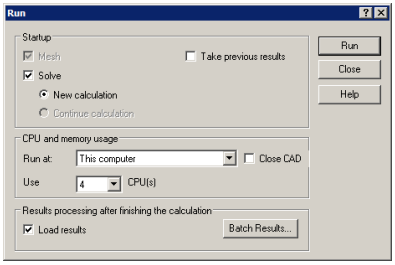
6 Make sure that **Pressure & stress** is selected in the **Dimensionality** list.

7 Click **OK** . The new **Equation Goal 1** item appears in the FloEFD Analysis Tree.

Running the Calculation

1 Click **Flow Analysis > Solve > Run**.

2 Click **Run**.



After the calculation has finished, close the **Monitor** dialog box.

Viewing the Goals

1 Right-click the **Goal Plots** icon under **Results** and select **Insert**.

2 Select the **Equation Goal 1** in the **Goals** dialog box.

3 Click **OK**.

An Excel spreadsheet with the goal results will open. The first sheet will contain a table presenting the final values of the goal.

You can see that the total pressure drop is about 120 Pa.

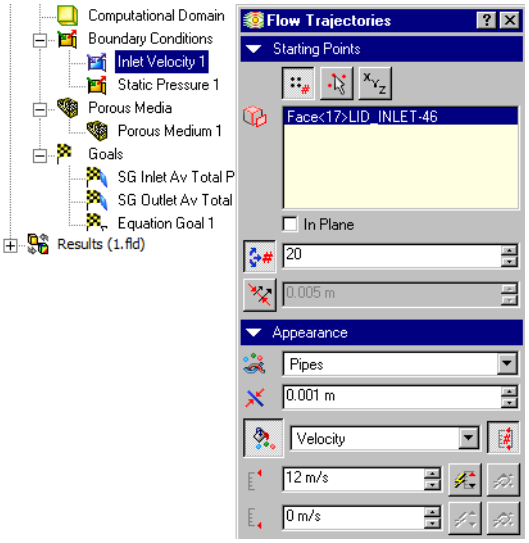
CATALYST.ASM [ISOTROPIC]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
Equation Goal 1	[Pa]	123,2906883	125,845	123,291	131,028	100	Yes

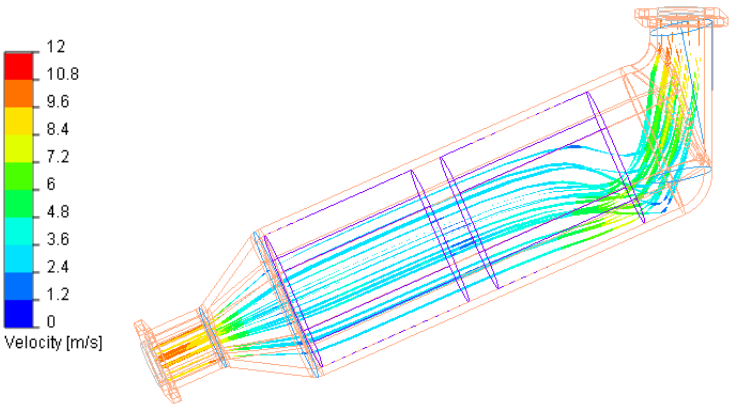
To see the non-uniformity of the mass flow rate distribution over a catalyst's cross section, we will display flow trajectories with start points distributed uniformly across the inlet.

Viewing Flow Trajectories

1. Right-click the **Flow Trajectories** icon and select **Insert**.
2. In the FloEFD Analysis Tree click the **Inlet Velocity 1** item. This selects the inner face of the inlet lid.
3. Under **Appearance**, make sure that **Color by Parameter** is selected, then change the parameter to **Velocity**.
4. Click **Adjust Minimum/Maximum and Number of Levels** and set the **Max** value to `12` m/s.
5. Click **OK**.
6. Select the **Hidden Line** view mode.



This is the picture you will see.



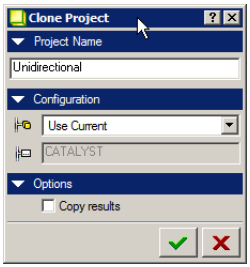
To compare the effectiveness of a unidirectional porous catalyst to an isotropic catalyst, let us calculate the project with a porous medium of unidirectional type.

Cloning the Project

1 Click **Flow Analysis > Project > Clone Project**.

2 Enter `Unidirectional` as the **Project name**.

3 Click **OK**.



Creating Unidirectional Porous Medium in the Engineering Database

The material you are going to create now is already defined in the **Engineering Database** under the **Pre-Defined** folder. You can skip this step and select the pre-defined "Unidirectional" material from the Engineering database when assigning the porous material to a component later in this tutorial.

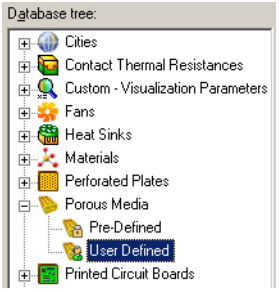
1 Click **Flow Analysis > Tools > Engineering Database**.

2 In the **Database tree** select **Porous Media / User Defined**.

3 On the **Items** tab select the **Isotropic** item**.**

4 Click **Copy** .

5 Click **Paste** . The new **Copy of Isotropic (1)** item appears in the list.

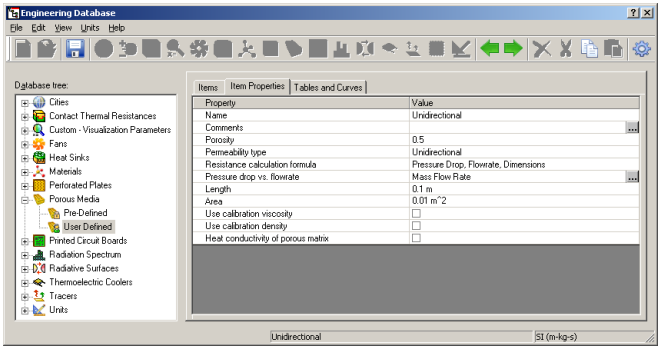


6 Select the **Copy of Isotropic (1)** item and click the **Item Properties** tab.

7 Rename the item to `Unidirectional`.

8 Change the **Permeability type** to **Unidirectional**.

9 Save the database and exit.



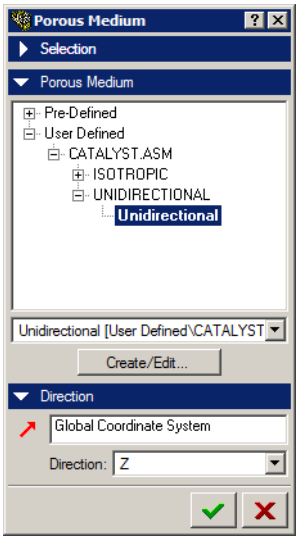
Now we can apply the new porous medium to the monoliths.

Specifying the Porous Medium - Unidirectional Type

1 Right-click the **Porous Medium 1** icon and select **Edit Definition**.

2 If you skipped the definition of the unidirectional porous medium, use the **Unidirectional** material available under **Pre-Defined**.

3 In the **Direction**, select the **Z** axis of the Global Coordinate System.



For porous media having unidirectional permeability, we must specify the permeability direction as an axis of the selected coordinate system (axis Z of the Global coordinate system in our case).

4 Click **OK**.

Since all other conditions and goals remain the same, we can start the calculation immediately.

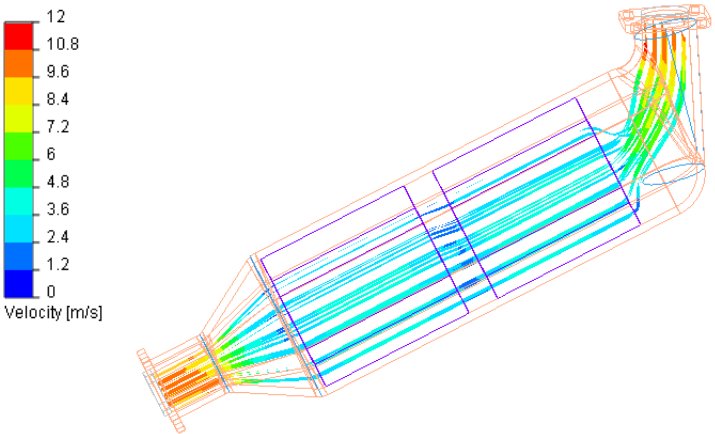
Comparing the Isotropic and Unidirectional Catalysts

When the calculation is finished, create the goal plot for the **Equation Goal 1**.

CATALYST.ASM [UNIDIRECTIONAL]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
Equation Goal 1	[Pa]	119,2174858	121,417	119,217	126,405	100	Yes

Display flow trajectories as described above.



Comparing the trajectories passing through the isotropic and unidirectional porous catalysts installed in the tube, we can summarize:

Due to the asymmetric position of the inlet tube with respect to the larger tube in which the catalyst bodies are installed, the incoming flow is non-uniform. Since the incoming flow is non-uniform, the flow inside the first catalyst body is non-uniform also. It is seen that the catalyst type (isotropic or unidirectional) affects both the incoming flow non-uniformity (slightly) and, more substantially, the flow within the catalysts (especially the first catalyst body). In both the cases the gas stream mainly enters the first catalyst body-closer to the wall opposite to the inlet tube. For the isotropic case, the gas flows into the first body nearer to the wall than for the case of the unidirectional catalyst. As a result, the flow in the initial (about one-third of the body length) portion of the first catalyst body is noticeably more non-uniform in the isotropic catalyst. Nevertheless, due to the isotropic permeability, the main gas stream expands in the isotropic catalyst and occupies a larger volume in the next part of the body than in the unidirectional catalyst, which, due to its unidirectional permeability, prevents the stream from expanding. So, the flow in the last two-thirds of the first catalyst body is less non-uniform in the isotropic catalyst. Since the distance between the two porous bodies installed in the tube is rather small, the gas stream has no time to become more uniform in the volume between the catalyst bodies, although in the unidirectional case a certain motion towards uniformity is perceptible. As a result, the flow non-uniformity occurring at the first catalyst body's exit passes to the second catalyst body. Then, it is seen that the flow non-uniformity does not change within the second catalyst body.

Let us now consider the flow velocity inside the catalyst. This is easy to do since the flow trajectories' colors indicate the flow velocity value in accordance with the specified palette. To create the same conditions for comparing the flow velocities in the isotropic and unidirectional catalysts, we have to specify the same velocity range for the palette in both the cases, since the maximum flow velocity governing the value range for the palette by default is somewhat different in these cases. It is seen that, considering the catalyst on the whole, the flow velocities in the isotropic and unidirectional catalysts are practically the same. Therefore, from the viewpoint of gas residence in the catalyst, there is no difference between the isotropic and unidirectional catalysts.

We can conclude that the isotropic catalyst is more effective than the unidirectional catalyst (of the same resistance to uniform flows), since the flow in it, as a whole, is more uniform. In spite of specifying the same resistance of the catalysts to flow, the overall pressure loss is lower by about 3% in the case of employing the unidirectional catalyst. This difference is due to the different flow non-uniformity both in the catalyst bodies and out of them.

B

Intermediate Examples

The **Intermediate Examples** presented below demonstrate how to solve engineering problems with FloEFD, using some of the most common tasks as examples.

B1 - Determination of Hydraulic Loss

B2 - Cylinder Drag Coefficient

B3 - Heat Exchanger Efficiency

B4 - Mesh Optimization

Intermediate Examples:

B1

Determination of Hydraulic Loss

In engineering practice the hydraulic loss of pressure head in any piping system is traditionally split into two components: the loss due to friction along straight pipe sections and the local loss due to local pipe features, such as bends, T-pipes, various cocks, valves, throttles, etc. Being determined, these losses are summed to form the total hydraulic loss. Generally, there are no problems in engineering practice to determine the friction loss in a piping system since relatively simple formulae based on theoretical and experimental investigations exist. The other matter is the local hydraulic loss (or so-called local drag). Here usually only experimental data are available, which are always restricted due to their nature, especially taking into account the wide variety of pipe shapes (not only existing, but also advanced) and devices, as well as the substantially complicated flow patterns in them. FloEFD presents an alternative approach to the traditional problems associated with determining this kind of local drag, allowing you to predict computationally almost any local drag in a piping system within good accuracy.

Opening the Model

Copy the **B1 - Hydraulic Loss** folder into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files. Run FloEFD.

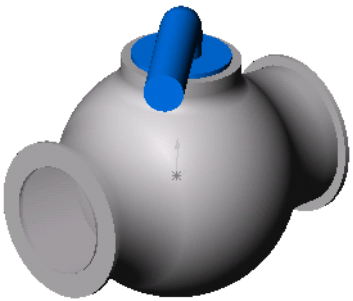
Click **File > Open**. In the **File Open** dialog box, browse to the **valve.asm** model located in the **B1 - Hydraulic Loss** folder and click **Open**.

In the **Select Instance** dialog box, select `40_DEGREES_SHORT_VALVE` and click **Open**.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **valve.asm** model located in the **B1 - Hydraulic Loss\Ready To Run** folder and run the desired projects.*

Model Description

This is a ball valve. Turning the handle closes or opens the valve.



The local hydraulic loss (or drag) produced by a ball valve installed in a piping system depends on the valve turning angle or on the minimum flow passage area governed by it. The latter depends also on a ball valve geometrical parameter, which is the ball-to-pipe diameter ratio governing the handle angle at which the valve becomes closed:

$$\theta = \arcsin\left[2\frac{D_{ball}}{D_{pipe}}\right]$$

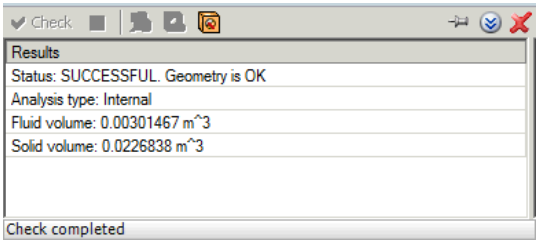
The standard engineering convention for determining local drag is by calculating the difference between the fluid dynamic heads measured upstream of the local pipe feature (ball valve in our case) and far downstream of it, where the flow has become uniform (undisturbed) again. In order to extract the pure local drag the hydraulic friction loss in the straight pipe of the same length must be subtracted from the measured dynamic head loss.

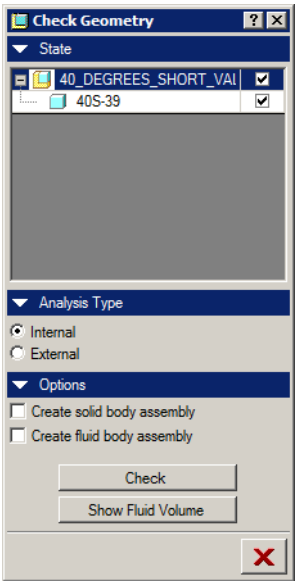
In this example we will obtain pressure loss (local drag) in the ball valve whose handle is turned by an angle of 40^o. The **Valve** analysis represents a typical FloEFD internal analysis.

📖 *Internal flow analyses deal with flows inside pipes, tanks, HVAC systems, etc. The fluid enters a model at the inlets and exits the model through outlets.*

To perform an internal analysis all the model openings must be closed with lids, which are needed to specify inlet and outlet flow boundary conditions on them. In any case, the internal model space filled with a fluid must be fully closed. You simply create lids as additional extrusions covering the openings. In this example the lids are semi-transparent allowing a view into the valve.

To ensure the model is fully closed click **Flow Analysis > Tools > Check Geometry**. Under **Analysis Type** select **Internal**. Then click **Check** to calculate the fluid and solid volumes of the model. If the fluid volume is equal to zero, the model is not closed.





Click **Show Fluid Volume** to see the volume that will be occupied by fluid in the analysis.

Close the **Check Geometry** dialog box.

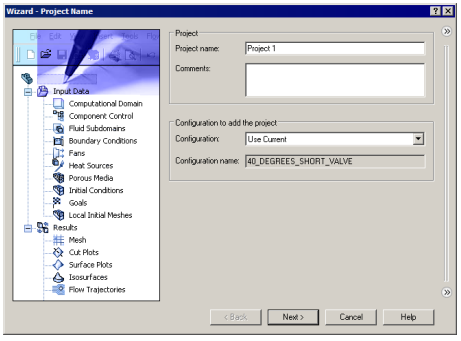
🕮 *The **Check Geometry** tool allows you to calculate the total fluid and solid volumes, check bodies for possible geometry problems (i.e. invalid contact) and visualize the fluid area and solid body as separate models.*

The first step is to create a new FloEFD project.

Creating a FloEFD Project

1 Click **Flow Analysis > Project > Wizard**. The project wizard guides you through the definition of a new FloEFD project.

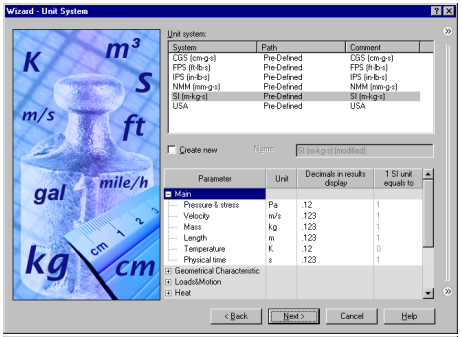
2 In the **Project Name** dialog box, type a new project name: `Project 1`.



🕮 *Each FloEFD project is associated with ainstance. You can attach the project either to the existing instance or create a new instance based on the current one.*

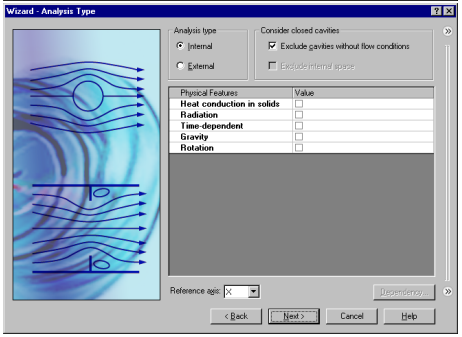
Click **Next**.

3 In the **Unit System** dialog box you can select the desired system of units for both input and output (results).
For this project use the International System **SI** by default.



Click **Next**.

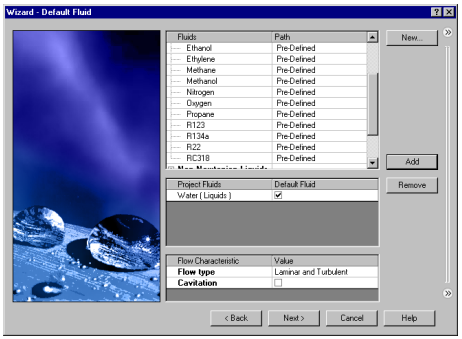
4 In the **Analysis Type** dialog box you can select either **Internal** or **External** type of the flow analysis.
To disregard closed internal spaces not involved in the internal analysis, you select **Exclude cavities without flow conditions**.
The **Reference axis of the global coordinate system** (X, Y or Z) is used for specifying data in a tabular or formula form in a cylindrical coordinate system based on this axis.



This dialog also allows you to specify advanced physical features you may want to take into account (heat conduction in solids, gravitational effects, time-dependent problems, surface-to-surface radiation, rotation).

Specify **Internal** type and accept the other default settings.
Click **Next**.

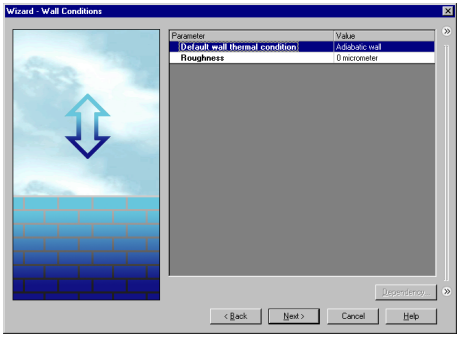
5 Since we use water in this project, open the **Liquids** folder and double-click the **Water** item.



📖 **Engineering Database** *contains numerical physical information on a wide variety of gas, liquid and solid substances as well as radiative surfaces. You can also use the Engineering Database to specify a porous medium. The Engineering Database contains pre-defined unit systems. It also contains fan curves defining volume or mass flow rate versus static pressure difference for selected industrial fans. You can easily create your own substances, units, fan curves or specify a custom parameter you want to visualize.*

Click **Next**.

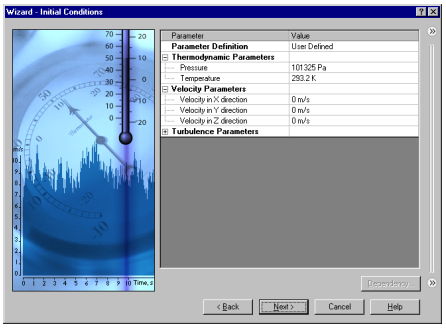
6 Since we do not intend to calculate heat conduction in solids, in the **Wall Conditions** dialog box you can specify the thermal wall boundary conditions applied by default to all the model walls contacting with the fluid.



For this project accept the default **Adiabatic wall** feature denoting that all the model walls are heat-insulated.

Click **Next**.

7 In the **Initial Conditions** dialog box specify initial values of the flow parameters. For steady internal problems, the specification of these values closer to the expected flow field will reduce the analysis convergence time.

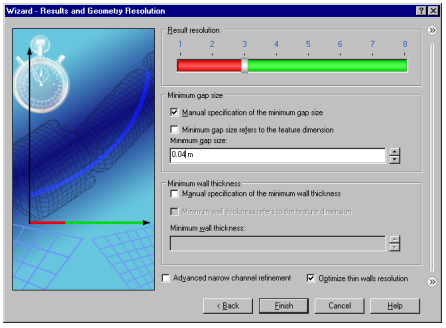


📖 *For steady flow problems FloEFD iterates until the solution converges. For unsteady (transient, or time-dependent) problems FloEFD marches in time for a period you specify.*

For this project use the default values.

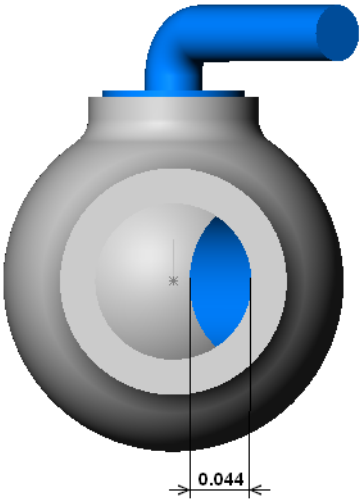
Click **Next**.

8 In the **Results and Geometry Resolution** dialog box you can control the analysis accuracy as well as the mesh settings and, through them, the required computer resources (CPU time and memory).


Wizard - Results and Geometry Resolution
Result resolution
1 2 3 4 5 6 7 8
Minimum gap size
Manual specification of the minimum gap size
Minimum gap size refers to the feature dimension
Minimum gap size:
0.04 m
Minimum wall thickness
Manual specification of the minimum wall thickness
Minimum wall thickness refers to the feature dimension
Minimum wall thickness:
Advanced narrow channel refinement
Optimize thin walls resolution
< Back
Finish
Cancel
Help

For this project accept the default result resolution `level 3`.

Select the **Manual specification of the minimum gap size** check box and enter `0.04` m for the **Minimum gap size**.



📖 *FloEFD calculates the default minimum gap size and minimum wall thickness using information about the overall model dimensions, the computational domain, and faces on which you specify conditions and goals. However, this information may be insufficient to recognize relatively small gaps and thin model walls. This may cause inaccurate results. In these cases, the Minimum gap size and Minimum wall thickness have to be specified manually.*

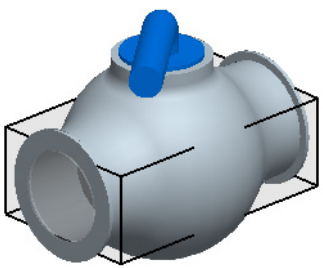
📖 **Result Resolution** *governs the solution accuracy via mesh settings and conditions of finishing the calculation that can be interpreted as resolution of calculation results. The higher the Result Resolution, the finer the mesh and the stricter the convergence criteria. Naturally, higher Result Resolution requires more computer resources (CPU time and memory).*
Geometry Resolution *(specified through the minimum gap size and the minimum wall thickness) governs proper resolution of geometrical model features by the computational mesh. Naturally, finer Geometry Resolution requires more computer resources.*

Click **Finish**.

📖 *The FloEFD Analysis tree provides a convenient specification of project data and view of results. You also can use the FloEFD Analysis tree to modify or delete the various FloEFD features.*

At the same time, a computational domain appears in the graphics window as a wireframe box.

📖 *The* **Computational Domain** *is a rectangular prism embracing the area inside which the flow and heat transfer calculations are performed.*



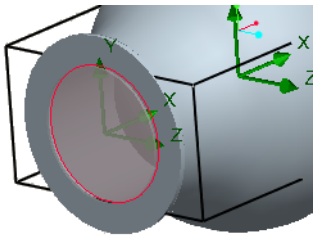
The next step is specifying **Boundary Conditions**. Boundary Conditions are used to specify fluid characteristics at the model inlets and outlets in an internal flow analysis or on model surfaces in an external flow analysis.

Specifying Boundary Conditions

1 Click **Flow Analysis > Insert > Boundary Condition**.

2 Select the inner face of the inlet lid.

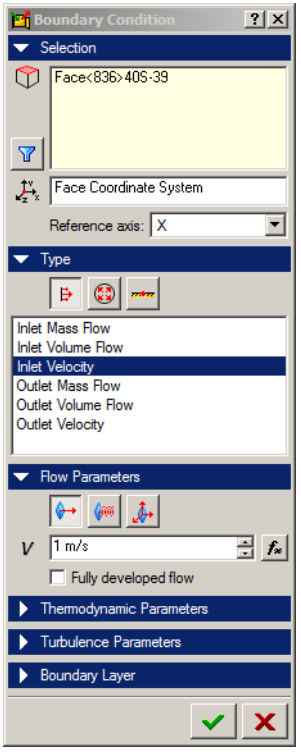
The selected face appears in the **Faces to Apply the Boundary Condition** list.



3 Under **Type** in the **Type of Boundary Condition** list, select the **Inlet Velocity** item.

4 Under **Flow Parameters**, click the **Velocity Normal to Face** edit box and set its value equal to `1 m/s` (type the value, the units will appear automatically).

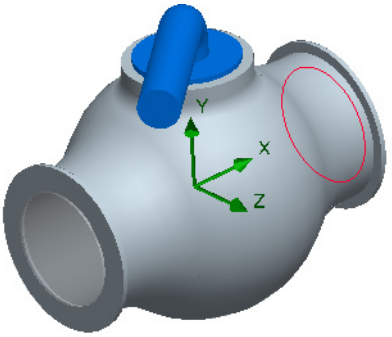
5 Accept all the other parameters and click **OK**.



This will simulate water flow entering the valve with the velocity of 1.0 m/s.

1 Select the inner face of the outlet lid.

2 Click **Flow Analysis > Insert > Boundary Condition**. The **Boundary Condition** dialog appears with the selected face in the **Faces to apply the boundary condition** list.



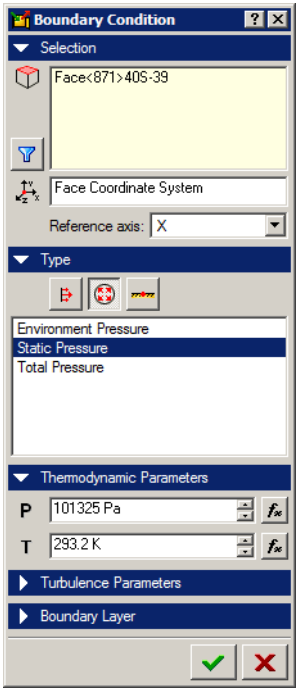
📖 *Before the calculation starts, FloEFD checks the specified boundary conditions for mass flow rate balance. The specification of boundary conditions is incorrect if the total mass flow rate on the inlets is not equal to the total mass flow rate on the outlets. In such case the calculation will not start. Also, note that the*

mass flow rate value is recalculated from the velocity or volume flow rate value specified on an opening. To avoid problems with specifying boundary conditions, we recommend that you specify at least one Pressure opening condition since the mass flow rate on a Pressure opening is automatically calculated to satisfy the law of conservation of mass.

3 Under **Type**, click **Pressure Openings** and in the **Type of Boundary Condition** list, select the **Static Pressure** item.

4 Under **Thermodynamic Parameters**, accept the default values for **Static Pressure** P (101325 Pa), **Temperature** T (293.2 K) and all the other parameters.

5 Click **OK**.



By specifying this condition we define that at the ball valve pipe exit the water has a static pressure of 1 atm.

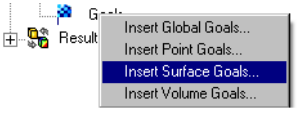
The hydraulic losses are calculated through the outlet and inlet total pressure difference ΔP from the following formula:

$$\xi = \frac{\Delta P}{\rho V^2/2}$$

where ρ is the water density, and V is water velocity. Since we already know the water velocity (specified by us as 1 m/s) and the water density (998.1934 kg/m^3 for the specified temperature of 293.2 K), then our goal is to determine the total pressure value at the valve's inlet and outlet. The easiest and fastest way to find the parameter of interest is to specify the corresponding engineering goal.

Specifying Surface Goals

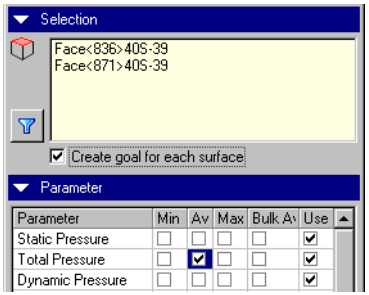
1 In the FloEFD Analysis tree, right-click the **Goals** icon and select **Insert Surface Goals.**



2 Select the inner faces of the inlet lid and the outlet lid (this can be done easily by holding down the **CTRL** key and clicking the corresponding boundary conditions in the FloEFD Analysis tree).

3 Select **Create goal for each surface** check box to create two separate goals, i.e. one for each of the selected faces.

4 In the **Parameter** table select the **Av** check box in the **Total Pressure** row.



5 Accept selected **Use for Conv.** check box to use the goals being created for convergence control.

6 Click **OK** . The new **SG Av Total Pressure 1** and **SG Av Total Pressure 2** items appear in the FloEFD Analysis tree.

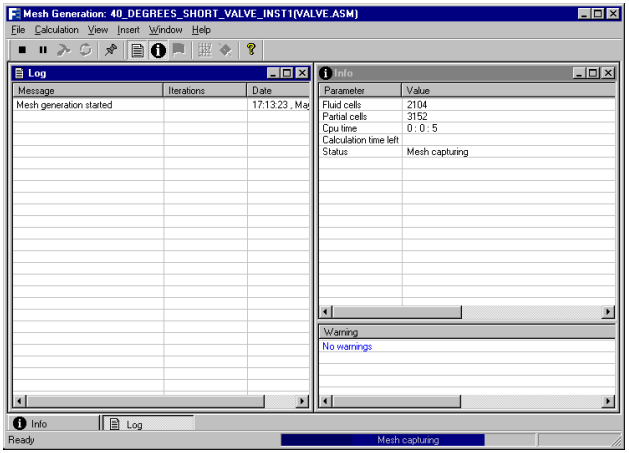
Now the FloEFD project is ready for the calculation. FloEFD will finish the calculation when the steady-state average value of total pressure calculated at the valve inlet and outlet are reached.

Running the Calculation

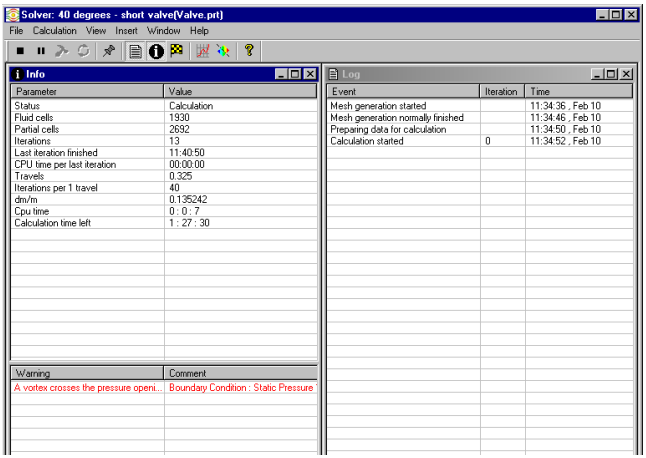
1 Click **Flow Analysis > Solve > Run**. The **Run** dialog box appears.

2 Click **Run** to start the calculation.

FloEFD automatically generates a computational mesh by dividing the computational domain into slices, which are further subdivided into cells. The cells are refined if necessary to resolve the model geometry properly. During the mesh generation process, you can see the current step in the **Mesh Generation** dialog box.



Monitoring the Solver


Solver: 40 degrees - short valve(Valve.prt)
File Calculation View Insert Window Help
Info
Parameter Value
Status Calculation
Fluid cells 1930
Partial cells 2692
Iterations 13
Last iteration finished 11:40:50
CPU time per last iteration 00:00:00
Travels 0.325
Iterations per 1 travel 40
dm/m 0.135242
Cpu time 0 : 0 : 7
Calculation time left 1 : 27 : 30
Warning Comment
A vortex crosses the pressure openi... Boundary Condition : Static Pressure
Log
Event Iteration Time
Mesh generation started 11:34:36 , Feb 10
Mesh generation normally finished 11:34:46 , Feb 10
Preparing data for calculation 11:34:50 , Feb 10
Calculation started 0 11:34:52 , Feb 10

After the calculation starts, the **Solver Monitor** dialog provides you with the current status of the solution. You can also monitor the goal changes and view preliminary results at selected planes
In the bottom pane of the **Info** window FloEFD notifies you if inappropriate results may occur. In our case, the message "**A vortex crosses the pressure opening**" appears to inform you that there is a vortex crossing the opening surface at which you specified the pressure boundary condition. In this case the vortex is broken into incoming and outgoing flow components. When flow both enters and exits an opening, the accuracy of the results is diminished. Moreover, there is no guarantee that convergence (i.e., the steady state goal) will be attained at all. Anyway, in case a vortex crosses a pressure opening the obtained results become suspect. If this warning persists we should stop the calculation and lengthen the ball valve outlet pipe to provide more space for development of the vortex. It is also expedient to attach the ball valve inlet pipe to avoid the flow disturbance caused by the valve's obstacle to affect the inlet boundary condition parameters.

Since the warning persists, click **File > Close** to terminate the calculation and exit the **Solver Monitor**.

You can easily extend the ball valve inlet and outlet sections by changing offset distance for the **DTM3** and **DTM4** features. Instead, we will simply clone the project to the pre-defined **40_DEGREES_LONG_VALVE** instance.

Cloning the Project

1 Click **Flow Analysis > Project > Clone Project**.

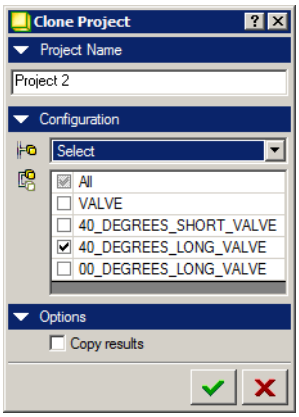
2 In the **Project Name**, type `Project 2.`

3 In the **Configuration to Add the Project** list, select **Select**.

4 In the **Configuration** list, select **40_DEGREES_LONG_VALVE**.

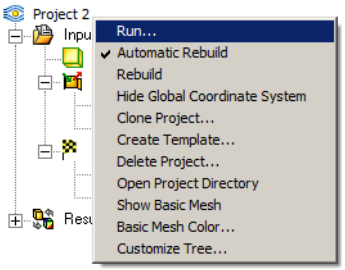
5 Click **OK**.

6 FloEFD has detected that the model was modified. Confirm the both warning messages with **Yes**.



The new FloEFD project, attached to the **40_DEGREES_LONG_VALVE** instance, has the same settings as the old one attached to the **40_DEGREES_SHORT_VALVE** so you can start the calculation immediately.

In the FloEFD Analysis tree, right-click the root **Project 2** item and select **Run**. Then click **Run** to start the calculation.

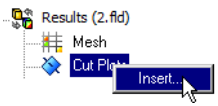


When the calculation is finished, close the **Solver Monitor** dialog box.

Let us now see the vortex notified by FloEFD during the calculation, as well as the total pressure loss.

Viewing Cut Plots

1 Right-click the **Cut Plots** icon and select **Insert**. The **Cut Plot** dialog box appears.



📖 *The **Cut Plot** displays results of a selected parameter in a selected view section. To define the view section, you can use planes or model planar faces (with the additional shift if necessary). The parameter values can be represented as a contour plot, as isolines, as vectors, or in a combination (e.g. contours with overlaid vectors).*

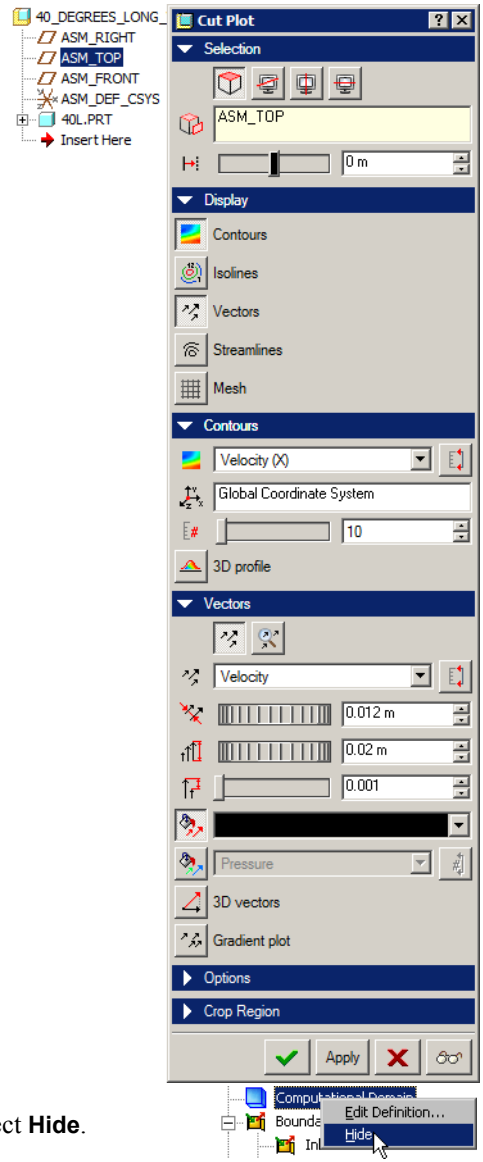
2 Click the Model Tree tab and select **ASM_TOP**. Its name appears in the **Section Plane or Planar Face** list.

3 In the **Cut Plot** dialog box under **Display**, in addition to displaying **Contours**, select **Vectors**.

4 Under **Contours**, select the **Velocity (X)** as the displayed parameter.

5 Under **Vectors**, set the **Vector Spacing** to 0.012 m and the **Arrow size** to 0.02 m.

6 Click **OK**.

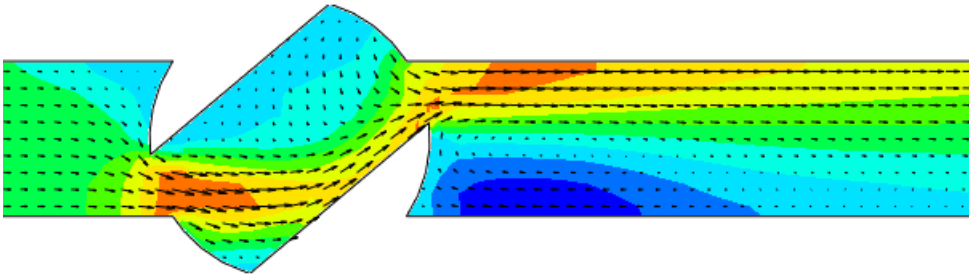


The new **Cut Plot 1** item appears in the FloEFD Analysis tree.

However, the cut plot cannot be seen through the non-transparent model. In order to see the plot, you can hide the model by clicking **Flow Analysis > Results > Display > Geometry**.

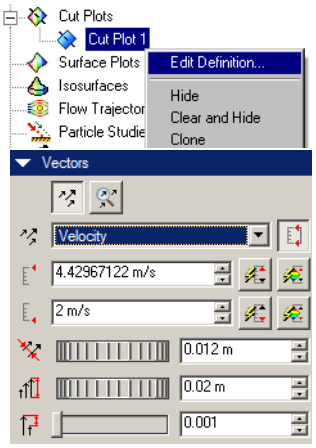
1 In the FloEFD Analysis tree, right-click the **Computational Domain** icon and select **Hide**.

Now you can see a contour plot of the velocity and the velocity vectors projected on the plot.



For better visualization of the vortex you can scale small vectors:

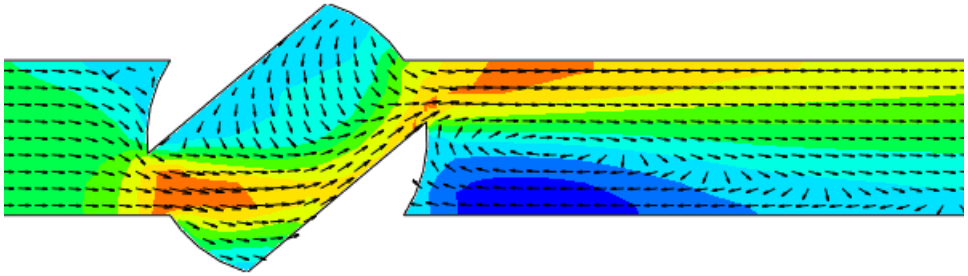
1 In the FloEFD Analysis tree, right-click the **Cut Plot 1** icon and select **Edit Definition**.



2 Under **Vectors** click the **Adjust Minimum and Maximum** and change the **Minimum** value to `2 m/s`.

*By specifying the custom **Minimum** we change the vector length range so that the vectors in areas where velocity is less than the specified **Minimum** value will appear as if it is equal to **Minimum**. This allows us to visualize the low velocity area in more detail.*

3 Click **OK** to save the changes and exit the **Cut Plot** dialog box. Immediately the cut plot is updated.



You can easily visualize the vortex by displaying the flow relative to the X axis. For that, you can display the x-component of velocity in a two-color palette and set the value, separating two colors, at zero.

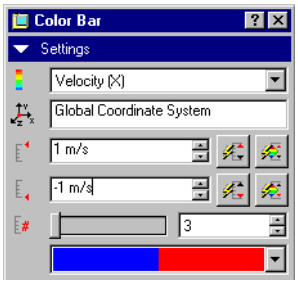
1 In the geometry area, double-click the palette bar or right-click on it and select **Edit**.

2 Under **Settings** using the slider set **Number of Levels** to **3**.

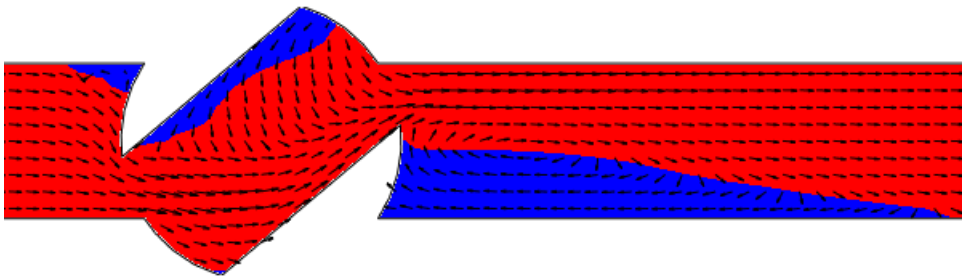
3 In the **Maximum** box type `1`.

4 In the **Minimum** box type `-1`.

5 Click **OK**.



Now the distribution of the **Velocity (X)** parameter is displayed in red-blue palette so that all the positive values are in red and all the negative values are in blue. This means that the blue area show the region of reverse flow, i.e. half of the vortex.

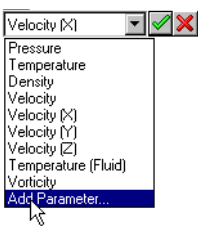


Next, we will display the distribution of total pressure within the valve.

Working with Parameter List

By default the total pressure is not included in the list of parameters available to display. To enable or disable a certain physical parameter for displaying, use **Customize Parameter List**.

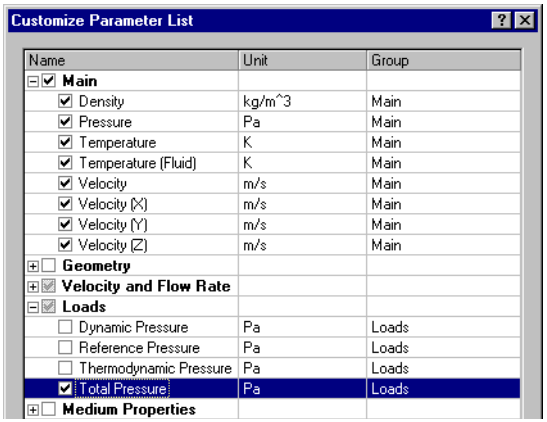
1 In the palette bar, click the caption with the name of the current visualization parameter and select **Add Parameter**.



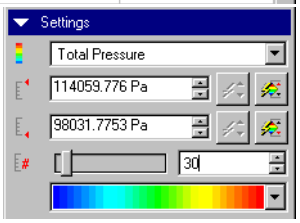
2 In the opened **Customize Parameter List** dialog box, expand the **Loads** group and select **Total Pressure**.

3 Click **OK** to save the changes.

4 In the geometry area double-click the palette bar. In the opened dialog box, change the visualization parameter to **Total Pressure**.

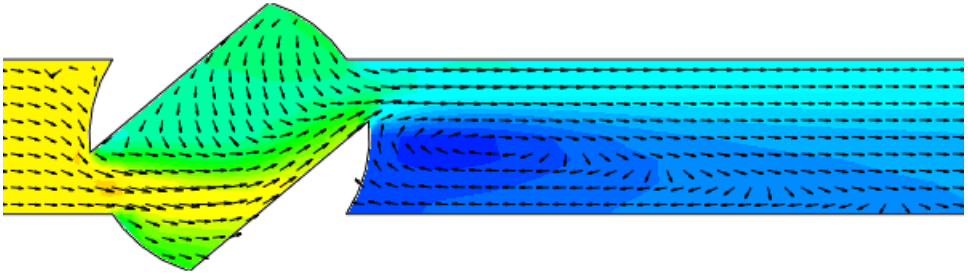


5 Under **Settings** using the slider, set the **Number of Levels** to about 30.



6 Click **OK** to save the changes and exit the **Color Bar** dialog box.

This will update the current cut plot to display the total pressure contour plot.



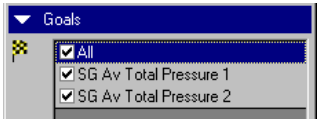
While the cut plot shows you the flow pattern, we will use the surface goal plot to determine the inlet and outlet values of total pressure required to calculate the loss.

Viewing the Goal Plot

📖 *The* **Goal Plot** *allows you to study how the goal value changed in the course of calculation. FloEFD uses Excel to display goal plot data. Each goal plot is displayed in a separate sheet. The converged values of all project goals are displayed in the* **Summary** *sheet of an automatically created Excel workbook.*

1 In the FloEFD Analysis tree, under **Results**, right-click the **Goal Plots** icon and select **Insert**. The **Goal Plot** dialog box appears.

2 Select **All**.



3 Click **OK** . The **Goals1** Excel workbook is created.

This workbook displays how the goal changed during the calculation. You can take the total pressure value presented at the **Summary** sheet.

VALVE.ASM [40_DEGREES_LONG_VALVE]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
SG Av Total Pressure 1	[Pa]	115177,1813	115186,177	115172,0662	115203,3627	100	Yes
SG Av Total Pressure 2	[Pa]	101841,635	101842,1902	101841,635	101844,0757	100	Yes

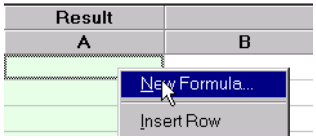
In fact, to obtain the pressure loss it would be easier to specify an Equation goal with the difference between the inlet and outlet pressures as the equation goal's expression. However, to demonstrate the wide capabilities of FloEFD, we will calculate the pressure loss with the FloEFD gasdynamic **Calculator**.

📖 *The* **Calculator** *contains various formulae from fluid dynamics, which can be useful for engineering calculations. The calculator is a very useful tool for rough estimations of the expected results, as well as for calculations of important characteristic and reference values. All calculations in the Calculator are performed only in the International system of units* **SI***, so no parameter units should be entered, and FloEFD Units settings do not apply in the Calculator.*

Working with Calculator

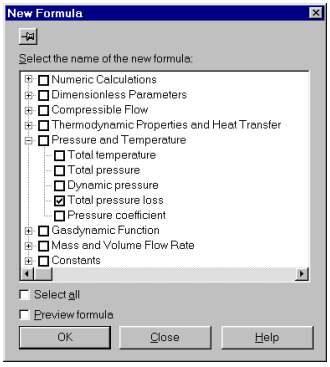
1 Click **Flow Analysis > Tools > Calculator**.

2 Right-click the A1 cell in the **Calculator** sheet and select **New Formula**. The **New Formula** dialog box appears.



3 In the **Select the name of the new formula** tree expand the **Pressure and Temperature** item and select the **Total pressure loss** check box.

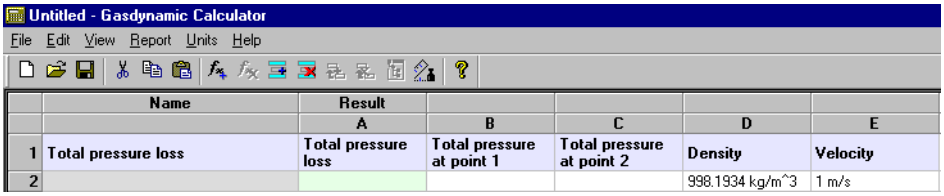
4 Click **OK**. The total pressure loss formula appears in the **Calculator** sheet.



In the **Result** (or **A**) column you see the formula name, in the next columns (**B**, **C**, etc.) you see names of the formula arguments (variables and constants). You can either type all the formula arguments' values in cells under their names in the SI units, or copy and paste them from the goals Excel worksheet table obtained through the **Goals** dialog box. The result value appears in the **Result** column cell immediately when you enter all the arguments and click another cell.

1 Specify the values in the cells as follows:

Density = 998.1934 (the water density at the specified temperature of 293.2 K), **Velocity** = 1.



	Name	Result				
		A	B	C	D	E
1	Total pressure loss	Total pressure loss	Total pressure at point 1	Total pressure at point 2	Density	Velocity
2					998.1934 kg/m^3	1 m/s

2 Open the **goals1** Excel workbook and copy the **Value** of **SG Av Total Pressure 1** into the Clipboard.

3 Go to the **Calculator**, click the **B2** cell and press **Ctrl+V** to paste the goal value from the Clipboard.

4 Return to Excel, copy the **Value** of **SG Av Total Pressure 2**. Go to the **Calculator,** click the **C2** cell and press **Ctrl+V**. Click any free cell. Immediately the **Total pressure loss** value appears in the **Result** column.

Total pressure loss	Total pressure loss	Total pressure at point 1	Total pressure at point 2	Density	Velocity
	26,71936	115177,2 Pa	101841,6 Pa	998.1934 kg/m^3	1 m/s

5 Click **File > Save**.

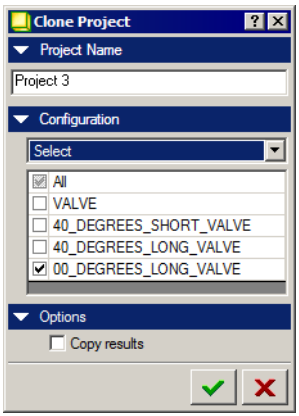
6 In the **Save As** dialog box browse to the folder where the ball valve model used in this example is located, enter `ball valve` for the file name, and click **Save**.

7 Click **File > Exit** to exit the **Calculator**.

To obtain the pure local drag, it is necessary to subtract from the obtained value the total pressure loss due to friction in a straight pipe of the same length and diameter. To do that, we perform the same calculations in the ball valve model with the handle in the 0^o angle position. You can do this with the **00_DEGREES_LONG_VALVE** configuration.

Since the specified conditions are the same for both **40_DEGREES_LONG_VALVE** and **00_DEGREES_LONG_VALVE** instance, it is useful to attach the existing FloEFD project to the **00_DEGREES_LONG_VALVE**instance.

Clone the current project to the **00_DEGREES_LONG_VALVE** instance.

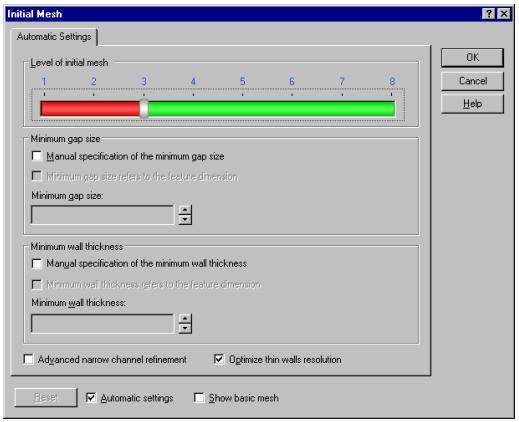


Since at zero angle the ball valve becomes a simple straight pipe, there is no need to set the **Minimum gap size** value smaller than the default gap size which, in our case, is automatically set equal to the pipe's diameter (the automatic minimum gap size depends on the characteristic size of the faces on which the boundary conditions are set). Note that using a smaller gap size will result in a finer mesh and, in turn, more computer time and memory will be required for calculation. To solve your task in the most effective way you should choose the optimal settings for the task.

Changing the Geometry Resolution

Check to see that the **Project 3** is the active project.

1 Click **Flow Analysis > Initial Mesh**.



*In Creo Parametric click **Flow Analysis > Mesh > Initial Mesh**.*

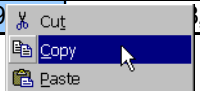
2 Clear the **Manual specification of the minimum gap size** check box.

3 Click **OK**.

Click **Flow Analysis > Solve > Run**. Then click **Run** to start the calculation.

After the calculation is finished, create the **Goal Plot**. The **goals2** workbook is created. Go to Excel, then select the both cells in the **Value** column and copy them into the Clipboard.

Goal Name	Unit	Value	Averaged Value	Minimum Value
SG Av Total Pressure 1	[Pa]	102028,2609	102039,6075	102024,2042
SG Av Total Pressure 2	[Pa]	101829	,9696	101827,9773



Now you can calculate the total pressure loss in a straight pipe.

1 Click **Flow Analysis > Tools > Calculator**.

2 In the **Calculator** menu, click **File > Open**. Browse to the folder where you saved the calculator file earlier in this tutorial and select the **ball valve.fwc** file. Click **Open**.

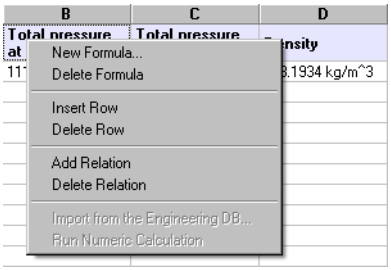
3 Click the **B4** cell and in the Calculator toolbar click to paste data from the Clipboard.

4 Save the existing value of the total pressure loss: click the **A2** cell, click , then click the **A7** cell and finally click .

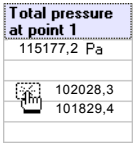
5 Double-click the **Name7** cell and type `40 degrees`.

Total pressure loss	Total pressure loss	Total pressure at point 1
	26,71936	115177,2 Pa
		102028,3
		101829,4
40 degrees	26,71936	

6 Right-click the **Total pressure at point 1** cell and select **Add Relation**. The cursor appears.

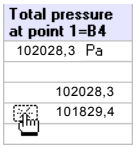


7 Click the **B4** cell. The value of total pressure is now taken from the **B4** cell.



8 Right-click the **Total pressure at point 2** cell and select **Add Relation**.

9 Click the **B5** cell. The value of total pressure is now taken from the **B5** cell. Immediately the total pressure value is recalculated.



Now you can calculate the local drag in the ball valve with the handle set at 40^{o}.

Total Pressure loss (40 deg)	Total Pressure loss (0 deg)	Local Drag
26,72	0,40	26,32

B2

Cylinder Drag Coefficient

Problem Statement

FloEFD can be used to study flow around objects and to determine the resulting lift and drag forces on the objects due to the flow. In this example we use FloEFD to determine the drag coefficient of a circular cylinder immersed in a uniform fluid stream. The cylinder axis is oriented perpendicular to the stream.

The computations are performed for a range of Reynolds numbers ($1,1000,10^5$), where $\mathrm{Re}=\frac{\rho UD}{\mu}$, D is the cylinder diameter, U is the velocity of the fluid stream, ρ is the density, and μ is the dynamic viscosity. The drag coefficient for the cylinder is defined as:

$$C_D = \frac{F_D}{\frac{1}{2}\rho U^2 DL}$$

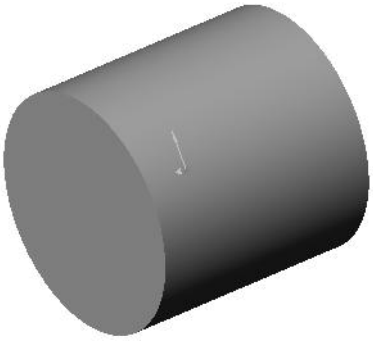
where F_D is the total force in the flow direction (i.e. drag) acting on a cylinder of diameter D and length L.The goal of the simulation is to obtain the drag coefficient predicted by FloEFD and to compare it to the experimental data presented in **Ref.1**.

Opening the Model

Copy the **B2 - Drag Coefficient** folder into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files. Run **FloEFD.**Click **File > Open**. In the **File Open** dialog box, browse to the **cylinder_001m.asm** assembly located in the **B2 - Drag Coefficient\cylinder 0.01m** folder and click **Open**.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **cylinder_001m.asm** assembly located in the **B2 - Drag Coefficient\cylinder 0.01m\Ready To Run** or the **cylinder_1m.asm** assembly **B2 - Drag Coefficient\cylinder 1m\Ready To Run** folder and run the desired projects.*

The **Cylinder** problem considered here represents a typical FloEFD **External** analysis.



📖 *External flows analyses deal with flows over or around a model such as flows over aircrafts, automobiles, buildings, etc. For external flow analyses the far-field boundaries are the Computational Domain boundaries. You can also solve a combined external and internal flow problem in a FloEFD project (for example flow around and through a building). If the analysis includes a combination of internal and external flows, you must specify External type for the analysis.*

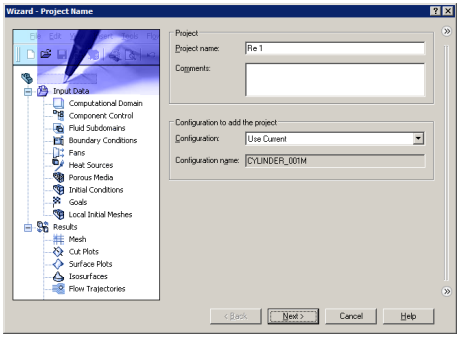
The first step is to create a new FloEFD project.

Creating a FloEFD Project

1 Click **Flow Analysis > Project > Wizard**. The project wizard guides you through the definition of a new FloEFD project. In this project we will analyze flow over the cylinder at the Reynolds number of 1.

2 In the **Project Name** dialog box, type a new project name: Re 1.

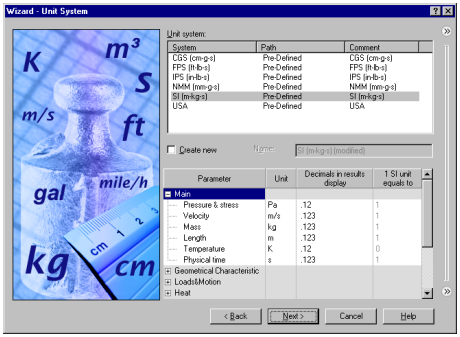
Click **Next**.



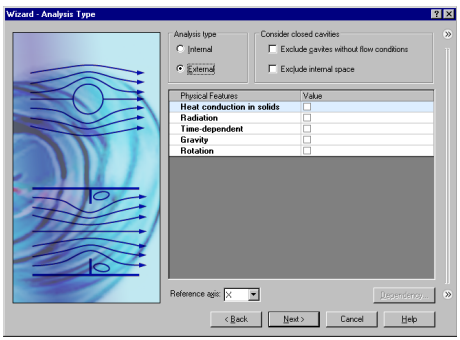
3 In the **Unit System** dialog box you can select the desired system of units for both input and output (results).

In this project we will specify the International System **SI** by default.

Click **Next**.



4 In the **Analysis Type** dialog box select an **External** type of flow analysis. This dialog also allows you to specify advanced physical features you want to include in the analysis. In this project we will not use any of the advanced physical features



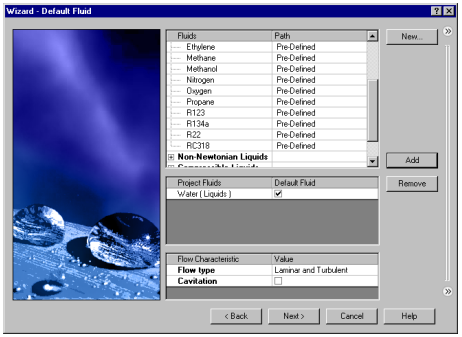
🕮 *To disregard closed internal spaces within the body you can select* ***Exclude internal spaces****; however no internal spaces exist within the cylinder in this tutorial. The* ***Reference axis of the global coordinate system*** *(X, Y or Z) is used for specifying data in a tabular or formula form with respect to a cylindrical coordinate system based on this axis.*

🕮 *The flow over a cylinder is steady at a Reynolds number Re < 40 (see the cylinder Re definition above) and unsteady (time-dependent) at Re > 40. Since in this tutorial the first calculation is performed at Re=1, to accelerate the run, we perform a steady-state analysis.*

Click **Next**.

5 Since we use water in this project, open the **Liquids** folder and double-click the **Water** item.

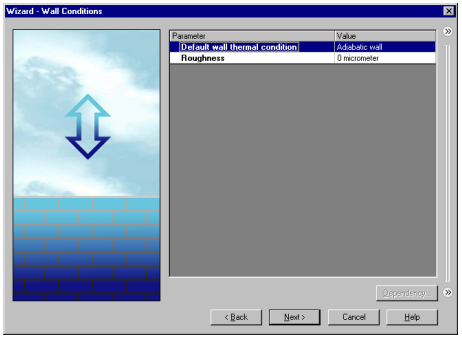
Click **Next**.



6 In the **Wall Conditions** dialog box you may specify the default thermal wall conditions applied to all the model walls in contact with the fluid.

In this project we keep the default **Adiabatic wall** setting, denoting that all the model walls are heat-insulated and accept the default zero wall roughness.

Click **Next**.

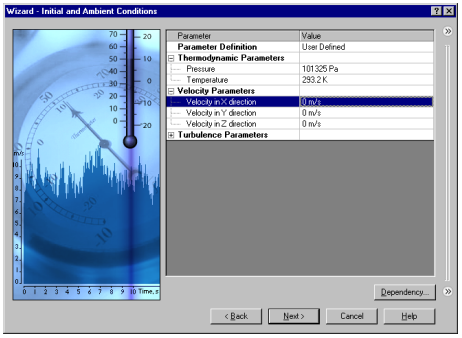


📖 *For a steady* **External** *problem, such as the cylinder in this tutorial, the* **Initial and Ambient Conditions** *dialog box asks you to specify the ambient flow conditions of the undisturbed free stream. Thus you will specify initial conditions inside the* **Computational Domain** *and boundary conditions at the* **Computational Domain** *boundaries. The ambient conditions are thermodynamic (static pressure and temperature by default), velocity, and turbulence parameters.*
In this project we consider the flow under the default thermodynamic conditions (i.e., the standard atmosphere at sea level), and set the incoming stream (X-component) velocity in accordance with the desired Reynolds number.

For convenience we can use the **Dependency** box to specify the incoming flow velocity in terms of the Reynolds number.

7 Click in the **Velocity in X direction** field. The **Dependency** button is enabled.

8 Click **Dependency**. The **Dependency** dialog box appears.



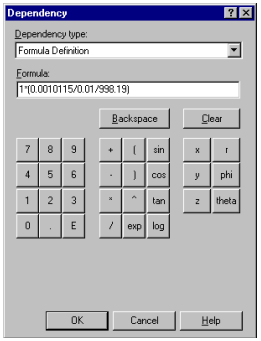
📖 *Using* ***Dependency*** *you can specify data in several ways: as a constant, as a tabular or formula dependency on x, y, z, r, θ, φ coordinates and time t (only for time-dependent analysis). The radius r is the distance from a point to the* ***Reference axis*** *selected from the reference coordinate system (the*

Global Coordinate System *for all data set in the* ***Wizard*** *and* ***General Settings*** *dialog boxes), while θ and φ are the polar and azimuthal angles of spherical coordinate system, respectively. Therefore, by combination of r, θ, and φ coordinates you can specify data in cylindrical or spherical coordinate systems.*

9 In the **Dependency type** list select **Formula Definition**.

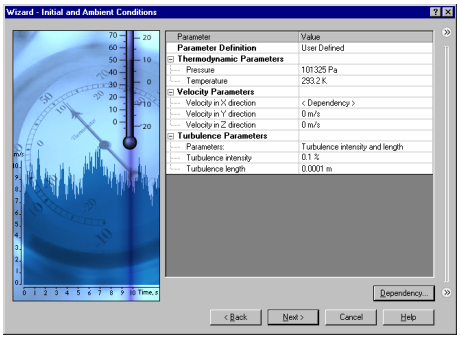
10 In the **Formula** box type the formula defining the flow velocity using the Reynolds number: `1*(0.0010115/0.01/998.19)`. Here:

1 – Reynolds number (Re)
0.0010115 (Pa*s) - water dynamic viscosity (μ) at the specified temperature of 293.2 K
0.01 (m) - cylinder diameter (D)
998.19 (kg/m^3)- water density (ρ) at the specified temperature of 293.2 K



11 Click **OK**. You will return to the **Initial and Ambient Conditions** dialog box.

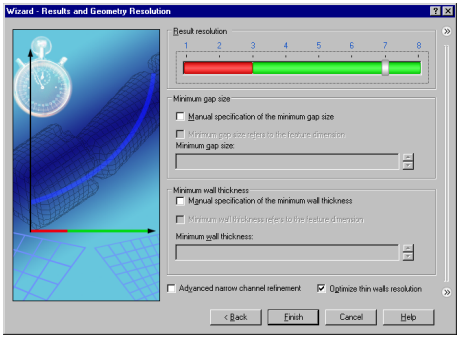
For most flows it is difficult to have a good estimation of their turbulence *a priori,* so it is recommended that the default turbulence parameters be used. The default turbulence intensity values proposed by FloEFD are 0.1% for external analyses and 2% for internal analyses and these values are appropriate for most cases. In this project we accept the default value of 0.1%.



Click **Next**.

12 In the **Result and Geometry Resolution** dialog box specify the result resolution level of **7** and accept the automatically defined minimum gap size and minimum wall thickness.

Click **Finish**. The project is created and the 3D Computational Domain is automatically generated.



In this tutorial we are interested in determining only the drag coefficient of the cylinder, without accounting 3D effects. Thus, to reduce the required CPU time and computer memory, we will perform a two-dimensional (2D) analysis in this tutorial.

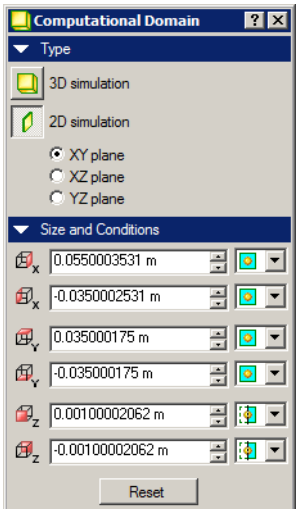
Specifying 2D simulation

1 In the FloEFD Analysis tree, expand the **Input Data** item.

2 Right-click the **Computational Domain** icon and select **Edit Definition**.

3 Under **Type** select **2D simulation** and **XY plane** (since the Z-axis is the cylinder axis).

4 Automatically the **Symmetry** condition is specified at the **Z min** and **Z max** boundaries of the Computational domain under **Size and Conditions**.

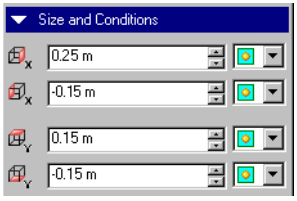


You can see that the **Z min** and **Z max** boundaries are set automatically, basing on the model dimensions.

Thus the reference cylinder length L in the cylinder drag (C_D) formula presented above is equal to $L = Z\ max - Z\ min = 0.002\ m$.

*For most cases, to study the flow field around an external body and to investigate the effects of design changes it is recommended to use the default **Computational Domain** size as determined by FloEFD. However, in this case we will compare the FloEFD results to experimental results and we would like to determine the drag coefficient with a high degree of accuracy. In order to eliminate any disturbances of the incoming flow at the **Computational Domain** boundaries due to the presence of the cylinder, we will manually set the boundaries farther away from the cylinder. The accuracy will be increased at the expense of required CPU time and memory due to the larger size of **Computational Domain**.*

5 Under **Size and Conditions** specify the X and Y coordinates of the Computational domain boundaries as shown on the picture to the right.



6 Click **OK** .

Since the incoming flow is aligned with the X-axis direction, the cylinder drag coefficient is calculated through the X-component of the force acting on the cylinder.

The X-component of force can be determined easily by specifying the appropriate FloEFD goal. In this case you will need to specify the **Force (X)** as a **Global Goal**. This ensures that the calculation will not be finished until **Force (X)** is fully converged in the entire computational domain (i.e. on the cylinder surface).

Specifying a Global Goal

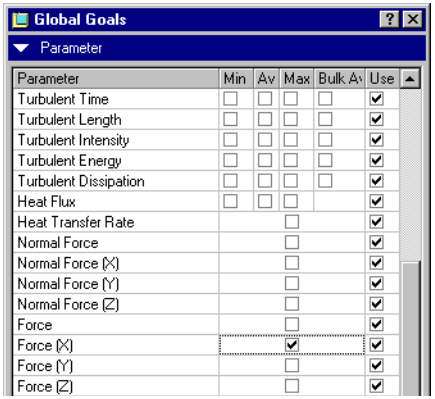
1 Click **Flow Analysis > Insert > Global Goals**.

*In Creo Parametric click **Flow Analysis > Insert > Goals > Global Goals**.*

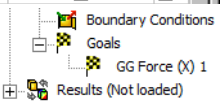
2 In the **Parameter** table select the first check box in the **Force (X)** row.

3 Accept selected **Use for Conv.** check box to use this goal for convergence control.

When choosing x, y, z-components of Force (or Torque) as goal you can select the Coordinate system in which these goals are calculated. In this example the default Global Coordinate System meets the task.



4 Click **OK** . The new **GG Force (X) 1** item appears in the FloEFD Analysis tree.



Specifying an Equation Goal

When the calculation is finished, you can manually calculate the drag coefficient from the obtained force value. Instead, let FloEFD make all the necessary calculations for you by specifying an **Equation Goal**.

1 Click **Flow Analysis > Insert > Equation Goal.**

*In Creo Parametric click **Flow Analysis > Insert > Goals > Equation Goal**.*

2 On the pane in the bottom of the screen click **Add Goal** .

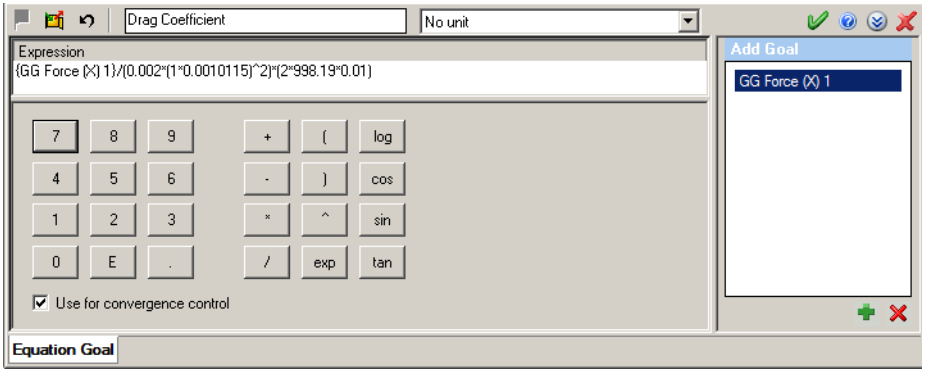
3 From the **Add Goal** list select the `GG Force (X) 1` goal and click **Add** . It appears in the **Expression** box.

4 Use buttons in the calculator or keyboard to complete the expression as follows:

```
{GG Force (X) 1}/(0.002*(1*0.0010115)^2)*(2*998.19*0.01).
```

5 Select **No unit** in the **Dimensionality** list.

6 Type `Drag Coefficient` in the **Goal Name** box.



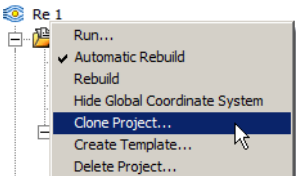
7 Click **OK** . The new equation goal appears in the FloEFD Analysis tree.

To compare the FloEFD results with the experimental curve taken from **Ref.1**, we will perform calculations considering Reynolds number values of 1, 10^3 and 10^5. As with Re = 1, the **cylinder_001m.asm** will be used to calculate the flow at the Reynolds number of 10^3. The **cylinder_1m.asm** will be used to calculate the flow at the Reynolds number of 10^5.

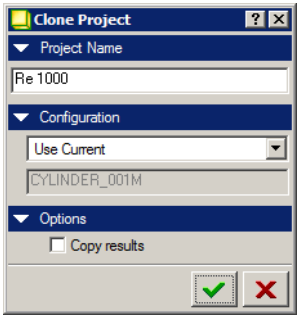
Cloning the Project

1 In the FloEFD Analysis tree, right-click the top **Re 1** icon and select **Clone Project**.

2 In the **Project name** box, type `Re 1000`.



3 Click **OK**. Now FloEFD creates a new project with the FloEFD data attached.



Since the new project is a copy of the **Re 1** FloEFD project, you only need to change the flow velocity value in accordance with the Reynolds number of 1000. Use the **General Settings** dialog box to change the data specified in the **Wizard**, except the settings for **Units** and **Result and Geometry Resolution**.

🕮 *The **General Settings** always presents the current state of the project parameters. You can change **General Settings** to correct the settings made in the **Wizard** or to modify the project created with the FloEFD **Template** in accordance with the new project requirements.*

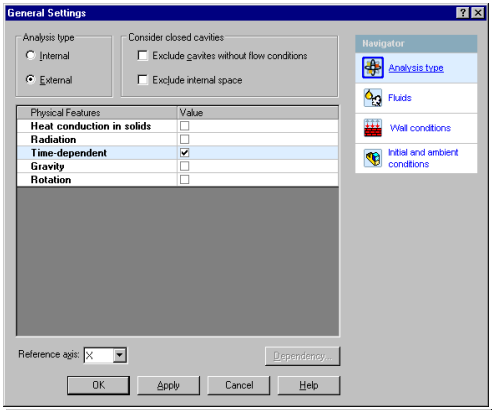
Changing Project Settings

1 Click **Flow Analysis > General Settings**. The **General Settings** dialog box appears.

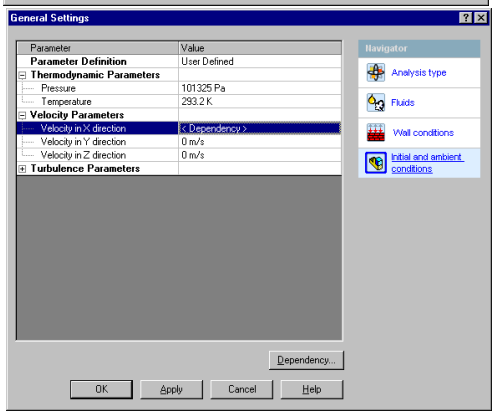
*In Creo Parametric click **Flow Analysis > Project > General Settings**.*

2 As it has been mentioned above, since the flow over a cylinder is unsteady at Re > 40, select the **Time-dependent** physical feature for this project.

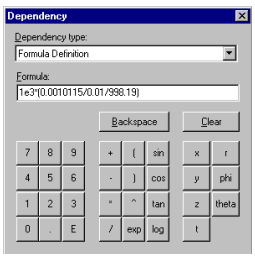
3 In the **Navigator** click **Initial and ambient conditions**.



4 Click the **Velocity in X direction** field and then click **Dependency**.



5 In the **Formula** box, type the formula for the new Reynolds number:
`1e3*(0.0010115/0.01/998.19).`



6 Click **OK** to return to the **General Settings** dialog box.

7 Click **OK** to save changes and close the **General Settings** dialog box.

Changing the Equation Goal

1 Right-click the **Drag Coefficient** icon under **Goals** and select **Edit Definition.**

2 In the **Expression** box type the new formula for the new Reynolds number:
`{GG Force (X) 1}/(0.002*(0.0010115*10^3)^2)*(2*998.19*0.01).`

3 Select **No unit** in the **Dimensionality** list.

4 Click **OK** to save changes and close the **Equation Goal** pane.

In the experiments performed with one fluid medium, the Reynolds number's large rise is usually obtained by increasing both the velocity and the model overall dimension (i.e. cylinder diameter) since it is difficult to increase only velocity by e.g. 10^5 times. Since our simulation is performed with water only, let us increase the cylinder diameter to 1 m to perform the calculation at a Reynolds number of 10^5.
Cloning a project is convenient if you want to create similar projects for the same model. The easiest way to apply the same general project settings to another model is to use the FloEFD **Template**.

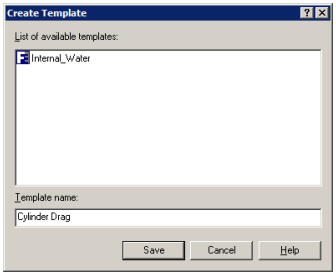
📖 ***Template*** *contains all of the general project settings that can be used as a basis for a new project. These settings are: problem type, physical features, fluids, solids, initial and ambient flow parameters, wall heat condition, geometry and result resolution, and unit settings. Notice that* ***Boundary Conditions***, ***Fans***, ***Initial Conditions***, ***Goals*** *and other features accessible from the* ***Flow Analysis > Insert*** *menu, as well as results are not stored in the template. Initially, only the Internal Water default template is available, but you can easily create your own templates*.

Creating a Template

1 Click **Flow Analysis > Project > Create Template**. The **Create Template** dialog box appears.

2 In the **Template name** box, type `Cylinder Drag`.

3 Click **Save**. The new FloEFD template is created.



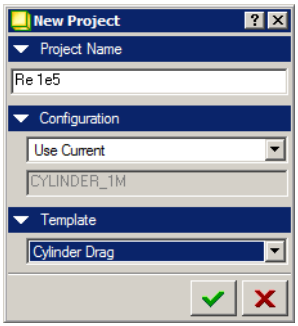
📖 *All templates are stored as* **.fwp** *files in the* **<install_dir>/Template** *folder, so you can easily apply a template to any previously created models.*

4 Save the model.

Next, create a new project based on the *Cylinder Drag* template.

Creating a Project from the Template

Open the **cylinder_1m.asm** file located in the **cylinder 1m** folder.



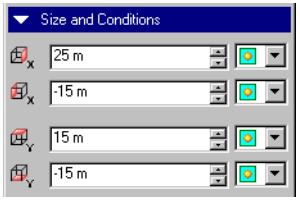
1 Click **Flow Analysis > Project > New**. The **New Project** dialog box appears.

2 In the **Project name** box, type `Re 1e5`.

3 In the **Project templates** list, select **Cylinder Drag**.

4 Click **OK**.

The newly created project has the same settings as the **Re 1000** project with the **cylinder_001m** model. The only exceptions are **Geometry Resolution** and **Computational Domain** size, which are calculated by FloEFD in accordance with the new model geometry.
Notice that the **2D simulation** setting and **Global Goal** are retained. Next, you can modify the project in accordance with the new model geometry.



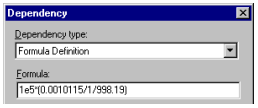
1 Click **Flow Analysis > Computational Domain** and adjust the computational domain size as shown at the picture to the right.

*In Creo Parametric click **Flow Analysis > Project > Computational Domain**.*

2 Click **OK** .

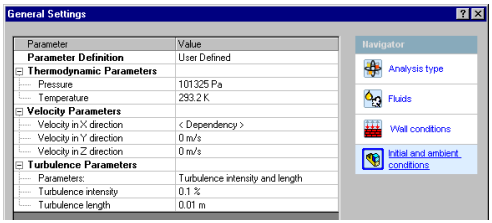
3 Open the **General Settings** dialog box and click **Initial and ambient conditions**, click the **Velocity in X direction** field, then click **Dependency**.

4 Change the velocity X component formula as follows: `1e5*(0.0010115/1/998.19)`.



Click **OK** to return to the General Settings dialog box.

By default, FloEFD determines the default turbulence length basis equal to one percent of the model overall dimension (i.e. cylinder diameter). Since the **Re 1e5** project was created from the template, it inherited the turbulence length value calculated for the small cylinder (d = 0.01m). For the **cylinder_1m** we need to change this value.



5 In the **General Settings** dialog box expand the **Turbulence parameters** item. Type `0.01` m in the **Turbulence length** field.

6 Click **OK**.

7 Create the **Equation Goal** for the drag coefficient of the cylinder as it was described before. In the Expression box enter the formula:

```
{GG Force (X) 1}/(0.2*(0.0010115*10^5)^2)*(2*998.19*1).
```

8 Select **No unit** in the **Dimensionality** list.

9 Type `Drag Coefficient` in the **Goal Name** box.

10 Click **OK** .

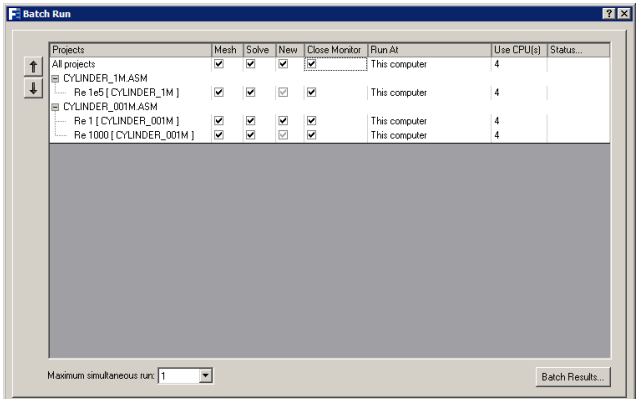
Now you can solve all of the projects created for both cylinders.

Solving a Set of Projects

FloEFD allows you to automatically solve a set of projects that exist in any currently opened document.

1 Click **Flow Analysis > Solve > Batch Run**.

2 Select the **Solve** check box in the **All projects** row to select **Solve** for all projects (**Re 1**, **Re 1000**, **Re 1e5**). Also select the **Close Monitor** check box in the **All projects** row. When the **Close Monitor** check box is selected, FloEFD automatically closes the **Solver Monitor** window when the calculation finishes.



3 Click **Run**.

Getting Results

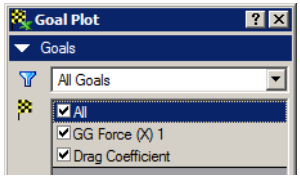
After all calculations are complete, activate the **Re 1000** project in the FloEFD projects tree. Create **Goal Plot** to obtain the **Drag Coefficient** value:

1 Click **Flow Analysis > Results** > **Load\Unload Results**.

2 In the **Load Results** dialog box, keep the default project's results file (2.fld) and click **Open**.

3 In the FloEFD Analysis tree, under **Results**, right-click the **Goal Plots** icon and select **Insert**. The **Goal Plot** dialog box appears.

4 Select **All**.



5 Click **OK** ✔. The **Goals1** Excel workbook is created. Switch to Excel to obtain the value.Activate the **Re 1** project and load results. Create the goal plot for both the goals.

CYLINDER_001M.ASM [RE1000]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value
GG Force (X) 1	[N]	0,000113488	0,000137279	0,000112818	0,000157575
Drag Coefficient	[ ]	1,107217709	1,339326042	1,100680396	1,537335639

CYLINDER_001M.ASM [RE1]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value
GG Force (X) 1	[N]	1,17543E-09	1,20537E-09	1,10834E-09	2,16893E-09
Drag Coefficient	[ ]	11,46775487	11,75983582	10,81316853	21,16057241

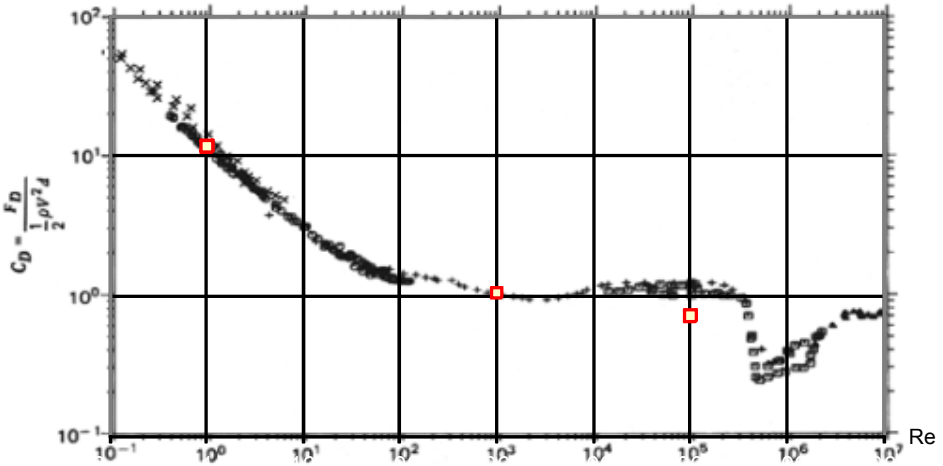
6 Switch to the **cylinder 1m** part, activate the **Re 1e5** project, load results and create the goal plot for both the goals.

CYLINDER_1M.ASM [RE1E5]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value
GG Force (X) 1	[N]	0,802145756	0,758909684	0,684053148	0,834139189
Drag Coefficient	[ ]	0,782590787	0,740408737	0,667377079	0,813804273

Even if the calculation is steady, the averaged value is more preferred, since in this case the oscillation effect is of less perceptibility. We will use the averaged goal value for the other two cases as well.

You can now compare FloEFD results with the experimental curve.



Ref. 1 Ronald L. Panton, "Incompressible flow" Second edition. John Wiley & sons Inc., 1995

B3

Heat Exchanger Efficiency

Problem Statement

FloEFD can be used to study the fluid flow and heat transfer for a wide variety of engineering equipment. In this example we use FloEFD to determine the efficiency of a counterflow heat exchanger and to observe the temperature and flow patterns inside of it. With FloEFD the determination of heat exchanger efficiency is straightforward and by investigating the flow and temperature patterns, the design engineer can gain insight into the physical processes involved thus giving guidance for improvements to the design.
A convenient measure of heat exchanger performance is its "efficiency" in transferring a given amount of heat from one fluid at higher temperature to another fluid at lower temperature. The efficiency can be determined if the temperatures at all flow openings are known. In FloEFD the temperatures at the fluid inlets are specified and the temperatures at the outlets can be easily determined. Heat exchanger efficiency is defined as follows:

$$\varepsilon = \frac{\textit{actual heat transfer}}{\textit{maximum possible heat transfer}}$$

The actual heat transfer can be calculated as either the energy lost by the hot fluid or the energy gained by the cold fluid. The maximum possible heat transfer is attained if one of the fluids was to undergo a temperature change equal to the maximum temperature difference present in the exchanger, which is the difference in the inlet temperatures of the hot and cold fluids, respectively: $(T_{hot}^{inlet} - T_{cold}^{inlet})$. Thus, the efficiency of a counterflow heat exchanger is defined as follows: $\varepsilon = \frac{T_{hot}^{inlet} - T_{hot}^{outlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}}$ - if hot fluid capacity rate is less than cold fluid capacity rate or $\varepsilon = \frac{T_{cold}^{outlet} - T_{cold}^{inlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}}$ - if hot fluid capacity rate is more than cold fluid capacity rate, where the capacity rate is the product of the mass flow and the specific heat capacity: $C = \dot{m}c$ (**Ref.2**)

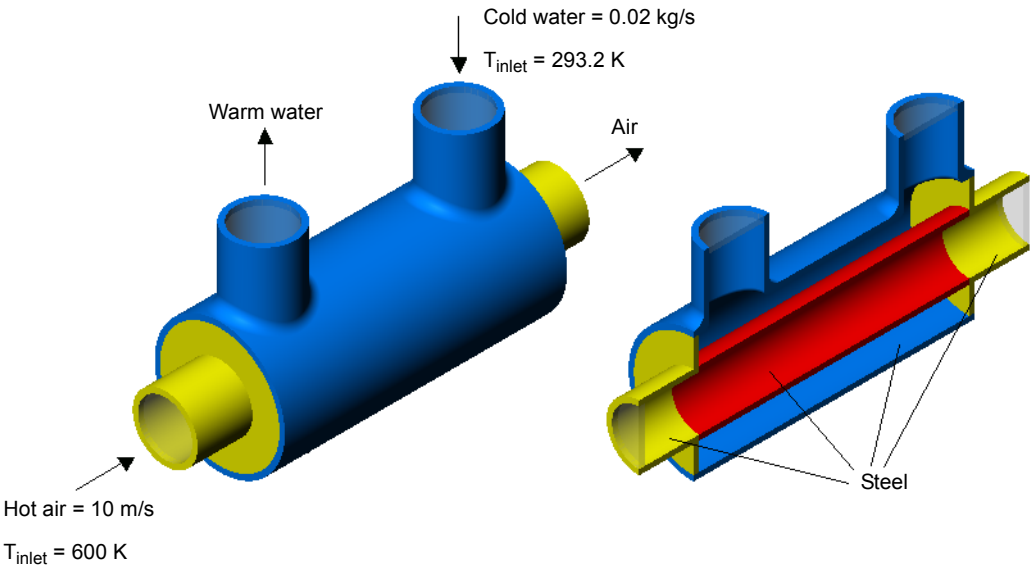
The goal of the project is to calculate the efficiency of the counterflow heat exchanger. Also, we will determine the average temperature of the heat exchanger central tube's wall. The obtained wall temperature value can be further used for structural and fatigue analysis.

Opening the Model

Copy the **B3 - Heat Exchanger** folder into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files. Run FloEFD.

Click **File > Open**. In the **File Open** dialog box, browse to the **heat_exchanger.asm** assembly located in the **B3 - Heat Exchanger** folder and click **Open** (or double-click the assembly).

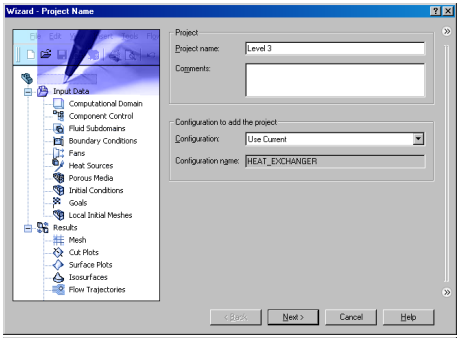
📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **heat_exchanger.asm** assembly located in the **B3 - Heat Exchanger\Ready To Run** folder and run the project.*



1 Click **Flow Analysis > Project**, **Wizard**.

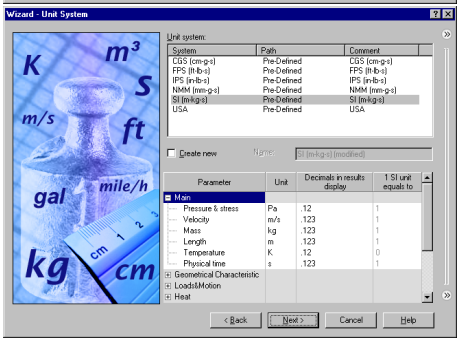
2 Type `Level 3` as the **Project name**. The 'Level_ 3' name was chosen because this problem will be calculated using **Result Resolution** level 3.

Click **Next**.

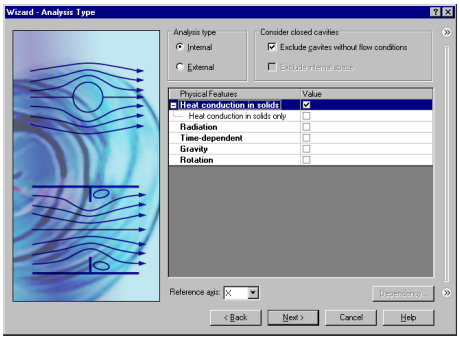


3 In the **Units** dialog box select the desired system of units for both input and output (results). For this project we will use the International System **SI** by default.

Click **Next**.



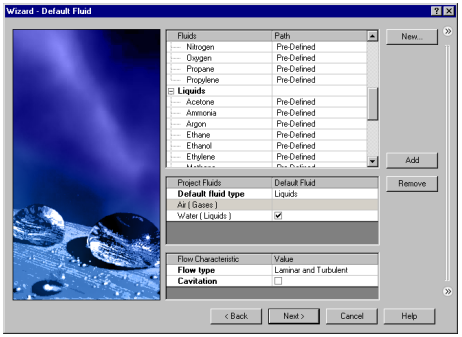
4 In the **Analysis Type** dialog box among **Physical features** select **Heat conduction in solids**.



🕮 *By default, FloEFD will consider heat conduction not in solids, but only within the fluid and between the walls and the fluid (i.e., convection). Selecting the* **Heat conduction in solids** *option enables the combination of convection and conduction heat transfer, known as conjugate heat transfer. In this project we will analyze heat transfer between the fluids through the model walls, as well as inside the solids.*

Click **Next**.

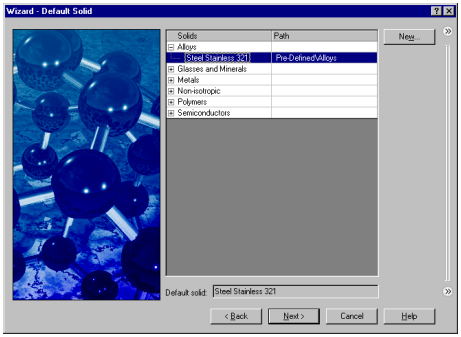
5 Since two fluids (water and air) are used in this project, expand the **Liquids** folder and add **Water** and then expand the **Gases** folder and add **Air** to the **Project Fluids** list. Check that the **Default fluid type** is **Liquids**.



Click **Next**.

6 Since we have selected the **Heat conduction in solids** option at the **Analysis Type** step of the Wizard, the **Default Solid** dialog box appears. In this dialog you specify the default solid material applied to all solid components. To assign a different material to a particular assembly component you need to create a **Solid Material** condition for this component.

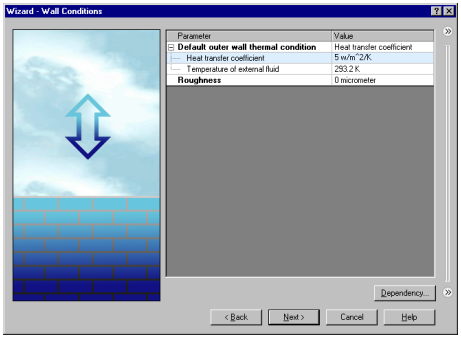
If the solid material you wish to specify as the default is not available in the **Solids** table, you can click **New** and define a new substance in the **Engineering Database**. The tube and its cooler in this project are made of stainless steel.



Expand the **Alloys** folder and click **Steel Stainless 321** to make it the default solid material.

Click **Next**.

7 In the **Wall Condition** dialog box, select **Heat transfer coefficient** as **Default outer wall thermal condition**.



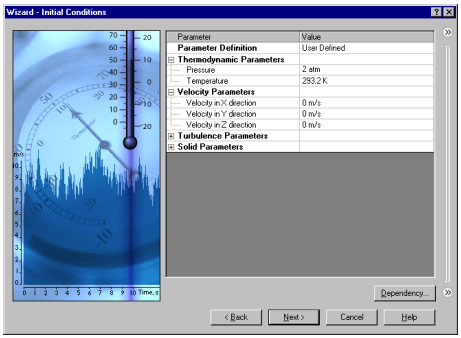
📖 *This condition allows you to define the heat transfer from the outer model walls to an external fluid (not modeled) by specifying the reference fluid temperature and the heat transfer coefficient value.*

Set the **Heat transfer coefficient** value to 5 $W/m^2/K$.

Leave the default (zero) wall roughness.

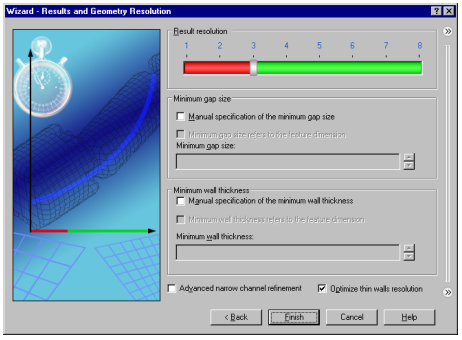
Click **Next**.

8 In the **Initial Conditions** dialog box under **Thermodynamics parameters** enter `2 atm` in the **Value** cell for the **Pressure** parameter. FloEFD automatically converts the entered value to the selected system of units.



Click **Next** accepting the default values of other parameters for initial conditions.

9 In the **Results and Geometry Resolution** dialog box we accept the default result resolution level 3 and the default minimum gap size and minimum wall thickness.



Click **Finish**.

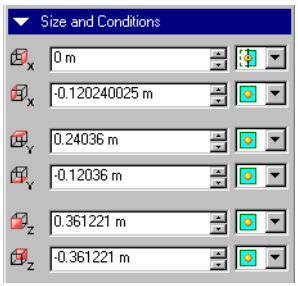
After finishing the **Wizard** you will complete the project definition by using the FloEFD Analysis tree. First of all you can take advantage of the symmetry of the heat exchanger to reduce the CPU time and memory required for the calculation. Since this model is symmetric, it is possible to “cut” the model in half and use a symmetry boundary condition at the plane of symmetry. This procedure is not required, but is recommended for efficient analyses.

Specifying Symmetry Condition

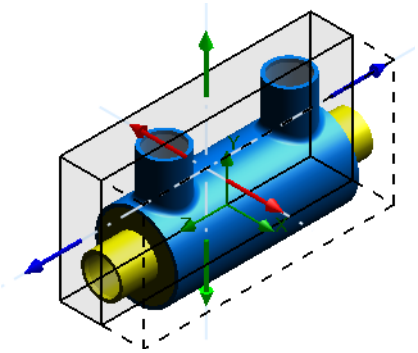
1 In the FloEFD Analysis tree, expand the **Input Data** item.

2 Right-click the **Computational Domain** icon and select **Edit Definition**.

3 Under **Size and Conditions** select the **Symmetry** condition at the **X max** boundary and type 0 in the **X max** box



📖 *To resize the domain manually, select the* **Computational Domain** *item in the FloEFD analysis tree, and in the graphics area click and drag the arrow handles at the sides of the computational domain frame to the desired positions, then adjust the exact coordinates in the appearing callouts..*



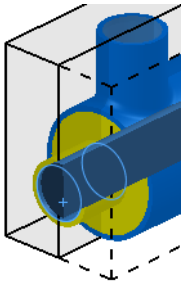
4 Click **OK** .

Specifying a Fluid Subdomain

Since we have selected **Liquids** as the **Default fluid type** and **Water** as the **Default fluid** in the Wizard, we need to specify another fluid type and select another fluid (air) for the fluid region inside the tube through which the hot air flows. We can do this by creating a **Fluid Subdomain**. When defining a **Fluid Subdomain** parameters we will specify **Gas** as the fluid type for the selected region, **Air** as the fluid and the initial temperature of 600 K and flow velocity of 10 m/s as the initial conditions in the selected fluid region.

1 Click **Flow Analysis > Insert > Fluid Subdomain**.

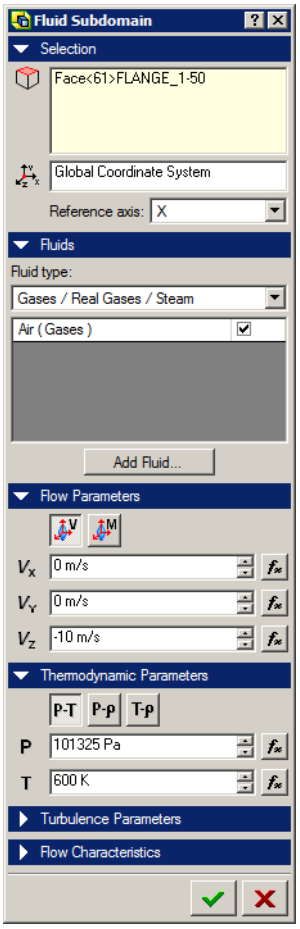
2 Select the **FLANGE_1** inner face (in contact with the fluid). Immediately the fluid subdomain you are going to create is displayed in the graphics area as a body of blue color.



📖 *To specify the fluid subdomain within a fluid region we must specify this condition on the one of the faces lying on the region's boundary - i.e. on the boundary between solid and fluid substances. The fluid subdomain specified on the region's boundary will be applied to the entire fluid region. You may check if the region to apply a fluid subdomain is selected properly by looking at the fluid subdomain visualization in the graphics area.*

3 Under **Selection**, accept the default **Coordinate System** and the **Reference axis**.

4 Under **Fluids** in the **Fluid type** list, select **Gases / Real Gases / Steam**. Because **Air** was defined in the Wizard as one of the **Project fluids** and you have selected the appropriate fluid type, it appears as the fluid assigned to the fluid subdomain.



🕮 *In the **Fluids** group box, FloEFD allows you to specify the fluid type and/or fluids to be assigned for the fluid subdomain as well as flow characteristics, depending on the selected fluid type.*

5 Under **Flow Parameters** in the **Velocity in Z Direction** box enter `-10`.

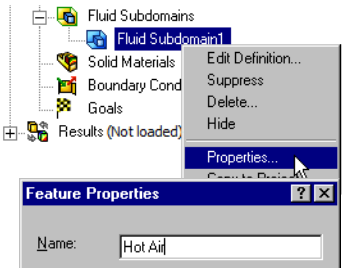
🕮 *FloEFD allows you to specify initial flow parameters, initial thermodynamic parameters, and initial turbulence parameters. These settings are applied to the specified fluid subdomain.*

6 Under **Thermodynamic parameters**, change the **Static Pressure** and **Temperature** to `1 atm` and 600 K respectively. FloEFD will automatically convert the entered values to the selected system of units.

These initial conditions are not necessary and the parameters of the hot air inlet flow are defined by the boundary condition, but we specify them to improve calculation convergence.

7 Click **OK**. The new **Fluid Subdomain1** item appears in the Analysis tree.

8 To easily identify the specified condition you can give a more descriptive name for the **Fluid Subdomain1** item. Right-click the **Fluid Subdomain1** item and select **Properties**. In the **Name** box type `Hot Air` and click **OK**.



🕮 *You can also click-pause-click an item to rename it directly in the FloEFD Analysis tree.*

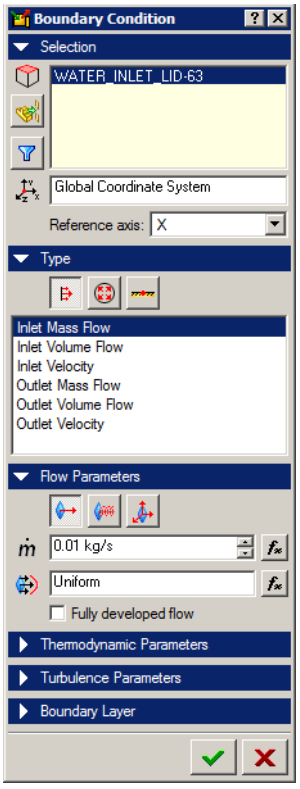
Specifying Boundary Conditions

1 Right-click the **Boundary Conditions** icon in the FloEFD Analysis tree and select **Insert Boundary Condition**. The **Boundary Condition** dialog box appears.

2 Select the **WATER_INLET_LID** component.

The selected component appears in the **Faces to Apply the Boundary Condition** list.

3 Under **Selection**, accept the default **Coordinate System** and **Reference axis**.



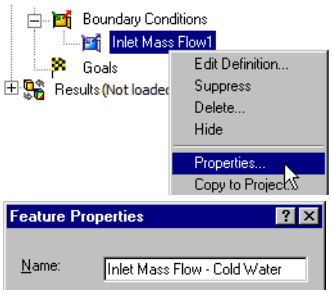
If you specify a new boundary condition, a callout showing the name of the condition and the default values of the condition parameters appears in the graphics area. You can double-click the callout to open the quick-edit dialog.

4 Under **Flow Parameters** in the **Mass Flow Rate** box, set the value equal to `0.01 kg/s`. Since the symmetry plane halves the opening, we need to specify a half of the actual mass flow rate.

5 Click **OK**. The new **Inlet Mass Flow 1** item appears in the Analysis tree.

This boundary condition specifies that water enters the steel jacket of the heat exchanger at a mass flow rate of 0.02 kg/s and temperature of 293.2 K.

6 Rename the **Inlet Mass Flow 1** item to `Inlet Mass Flow - Cold Water`.



Next, specify the water outlet **Environment Pressure** condition.

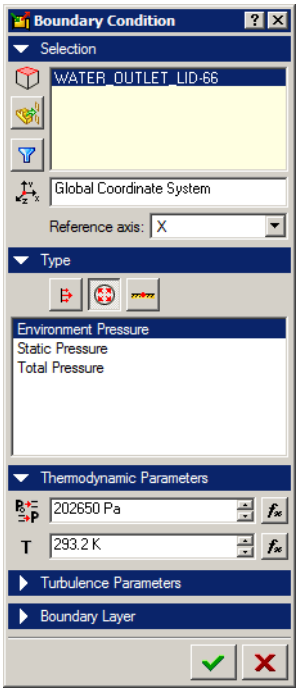
7 In the FloEFD Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

8 Select the **WATER_OUTLET_LID** component. The selected component appears in the **Faces to Apply the Boundary Condition** list.

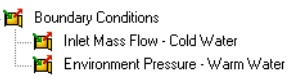
9 Under **Type**, click **Pressure Openings** and in the **Type of Boundary Condition** list select the **Environment Pressure** item.

10 Under **Thermodynamic Parameters**, accept the value of **Environment Pressure** (202650 Pa), taken from the value specified at the **Initial Conditions** step of the **Wizard**, the default value of **Temperature** T (293.2 K) and all other parameters.

11 Click **OK**. The new **Environment Pressure 1** item appears in the FloEFD Analysis tree.



12 Rename the **Environment Pressure 1** item to `Environment Pressure - Warm Water`.



Next we will specify the boundary conditions for the hot air flow.

13 In the FloEFD Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

14 Select the **AIR_INLET_LID** component.

The selected component appears in the **Faces to Apply the Boundary Condition** list. Accept the default **Coordinate System** and **Reference axis**.

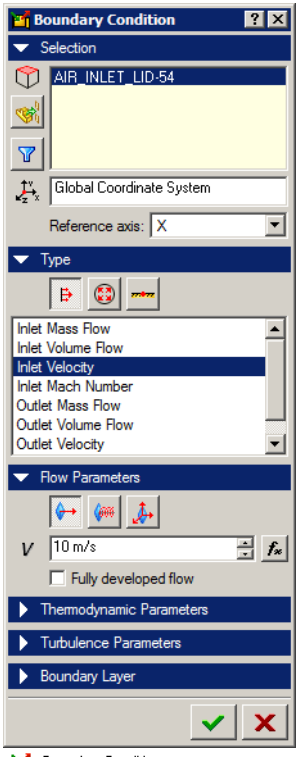
15 Under **Type**, select the **Inlet Velocity** condition.

16 Under **Flow Parameters** in the **Velocity Normal to Face** box, set the value equal to `10` (type the value, the units will appear automatically).

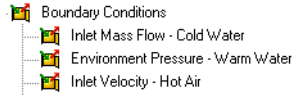
17 Expand the **Thermodynamic Parameters** item. The default temperature value is equal to the value specified as the initial temperature of air in the **Fluid Subdomain** dialog box. We accept this value.

18 Click **OK**. The new **Inlet Velocity 1** item appears in the Analysis tree.

This boundary condition specifies that air enters the tube at the velocity of 10 m/s and temperature of 600 K.



19 Rename the **Inlet Velocity 1** item to `Inlet Velocity - Hot Air`.

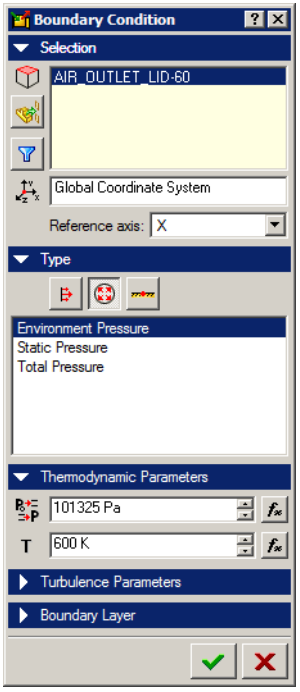


Next specify the air outlet **Environment Pressure** condition.

20 In the FloEFD Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**. The **Boundary Condition** dialog box appears.

21 Select the **AIR_OUTLET_LID** component.

The selected component appears in the **Faces to Apply the Boundary Condition** list.

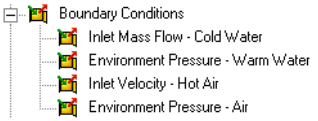


22 Under **Type**, click **Pressure Openings** and in the **Type of Boundary Condition** list select the **Environment Pressure** item.

23 Under **Thermodynamic Parameters**, make sure that the **Environment Pressure** and **Temperature** are set to 101325 Pa and 600 K respectively. Accept the default values of other parameters.

Click **OK**.

24 Rename the new item **Environment Pressure 1** to `Environment Pressure - Air`.


Boundary Conditions
Inlet Mass Flow - Cold Water
Environment Pressure - Warm Water
Inlet Velocity - Hot Air
Environment Pressure - Air

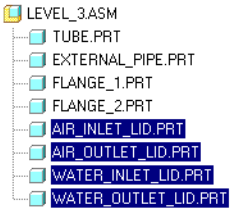
This project involving analysis of heat conduction in solids. Therefore, you must specify the solid materials for the model's components and the initial solid temperature.

Specifying Solid Materials

Notice that the auxiliary lids on the openings are solid. Since the material for the lids is the default stainless steel, they will have an influence on the heat transfer. You cannot suppress or disable them in the **Component Control** dialog box, because boundary conditions must be specified on solid surfaces in contact with the fluid region. However, you can exclude the lids from the heat conduction analysis by specifying the lids as insulators.

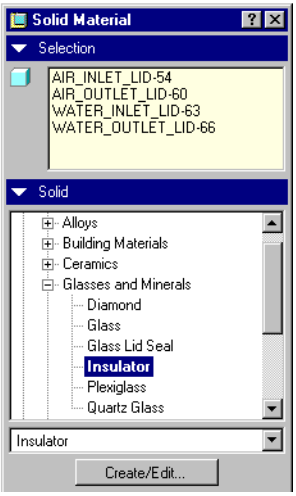
1 Right-click the **Solid Materials** icon and select **Insert Solid Material**.

2 In the Model Tree, select all the lid parts. As you select the lids, their names appear in the **Components to Apply the Solid Material** list.



3 In the **Solid** group box expand the list of **Pre-Defined** materials and select the **Insulator** solid in the **Glasses & Minerals** folder.

4 Click **OK** . Now all auxiliary lids are defined as insulators.



📖 *The thermal conductivity of the* **Insulator** *substance is zero. Hence there is no heat transferred through an insulator.*

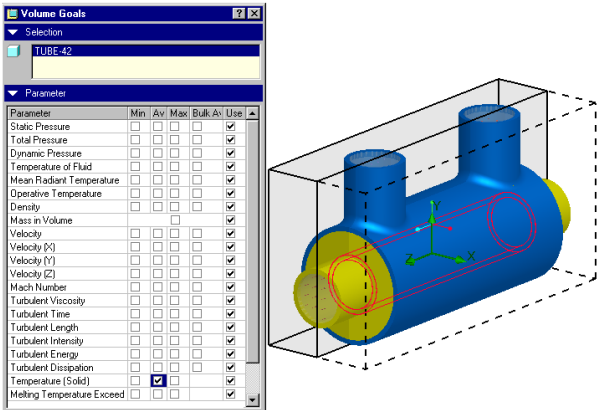
5 Rename the **Insulator Solid Material 1** item to `Insulators`.

Specifying a Volume Goal

1 In the FloEFD Analysis tree, right-click the **Goals** icon and select **Insert Volume Goals**.

2 In the Model tree select the **Tube** part.

3 In the **Parameter** table select the **Av** check box in the **Temperature (Solid)** row. Accept the selected **Use for Conv.** check box to use this goal for convergence control.



4 In the **Name template** type `VG Av T of Tube.`

5 Click **OK** .

Running the Calculation

1 Click **Flow Analysis > Solve > Run**. The **Run** dialog box appears.

2 Click **Run**.

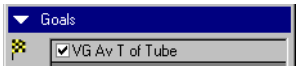
After the calculation finishes you can obtain the temperature of interest by creating the corresponding **Goal Plot**.

Viewing the Goals

In addition to using the FloEFD Analysis tree you can use FloEFD Toolbars to get fast and easy access to the most frequently used FloEFD features. Toolbars are very convenient for displaying results.

1 Click **Generate goal plot** on the toolbar. The **Goal Plot** dialog box appears.

2 Select the goals of the project (actually, in our case there is only one goal) .



3 Click **OK** . The **Goals1** Excel workbook is created.

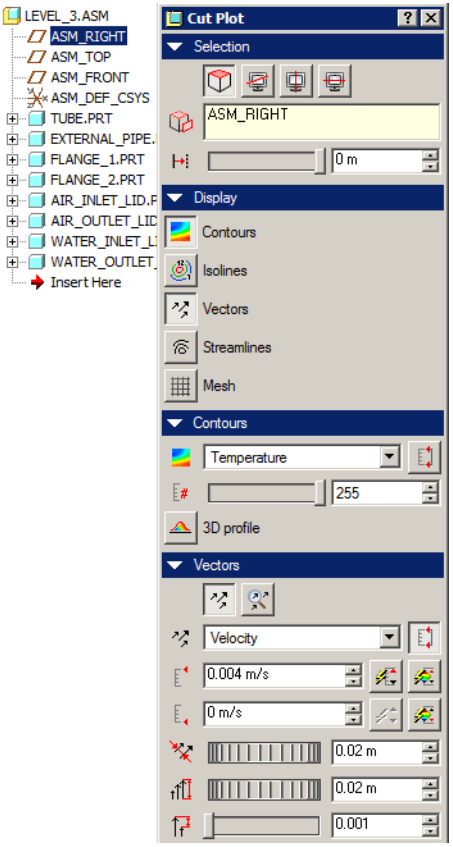
You can view the average temperature of the tube on the **Summary** sheet.

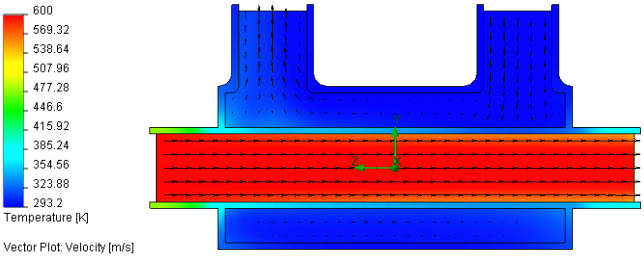
HEAT_EXCHANGER.ASM [LEVEL3]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
VG Av T of Tube	[K]	344,504625	343,93285	343,1591445	344,5243078	100	Yes

Viewing Cut Plots

1. Click **Cut Plot** . The **Cut Plot** dialog box appears.
2. In the Model Tree select **ASM_RIGHT**.
3. In the **Cut Plot** dialog under **Display**, in addition to displaying **Contours** , select **Vectors** .
4. Under **Contours** specify the parameter which values to show at the contour plot. In the **Parameter** box, select **Temperature**.
5. Using the slider set the **Number of Levels** to `255`.
6. Under **Vectors** make sure that **Static Vectors** is selected. Click the **Adjust Minimum and Maximum** and change the **Maximum** velocity to `0.004 m/s`.
7. Click **OK** .
8. The cut plot is created but the model overlaps it.
9. Click **Geometry** icon to hide the model.

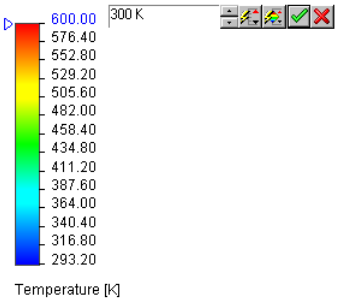


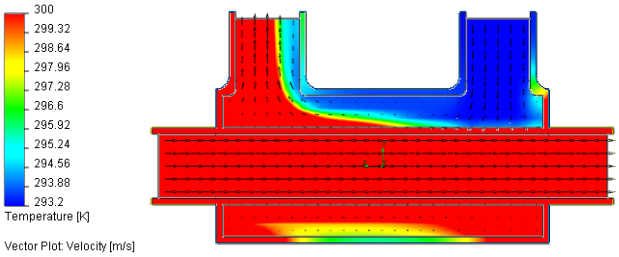


Adjusting the Parameter Display Range

1 In the temperature palette bar click the maximum value and type `300 K` in an edit box

2 Click . This will update the current cut plot in accordance with the specified temperature range.





To see how the water flows inside the exchanger we will display the **Flow Trajectories**.

Let us now display how the flow develops inside the exchanger.

In the FloEFD Analysis tree right-click the **Cut Plot 1** icon and select **Hide**.

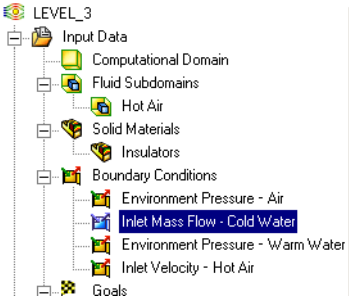
3 Click **Geometry** to show the model, then on the **Model Display** toolbar click **Hidden Line** to show the face outlines.

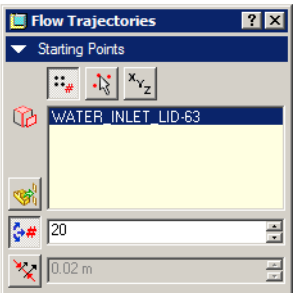
Displaying Flow Trajectories

1 Click **Flow Trajectories** . The **Flow Trajectories** dialog appears.

2 In the FloEFD Analysis tree select the **Inlet Mass Flow – Cold Water** item.

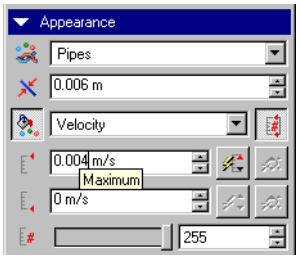
This will select the inner face of the **WATER_INLET_LID** to place the trajectories start points on it.





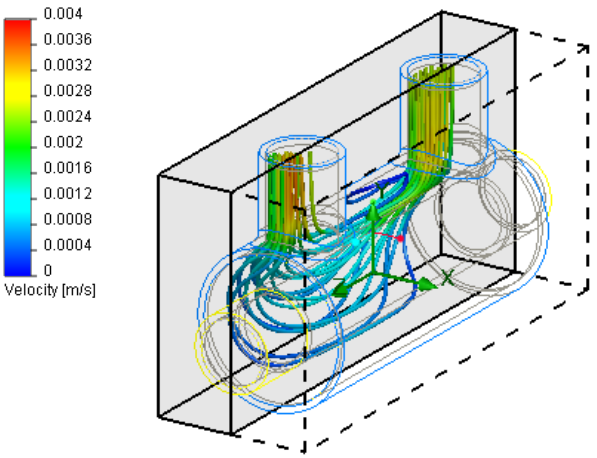
3 Under **Appearance**, in the **Color by Parameter** list, select **Velocity**.

4 Click the **Adjust Minimum/Maximum and Number of Levels** and set **Maximum** velocity to `0.004 m/s`.



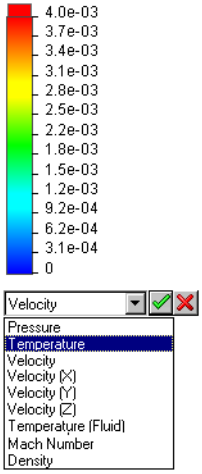
5 Click **OK** . Trajectories are created and displayed.

By default the trajectories are colored in accordance with the distribution of the parameter specified in the **Color by Parameter** list. Since you specified velocity, the trajectory color corresponds to the velocity value. To define a fixed color for flow trajectories click **Color** and select a desired color.

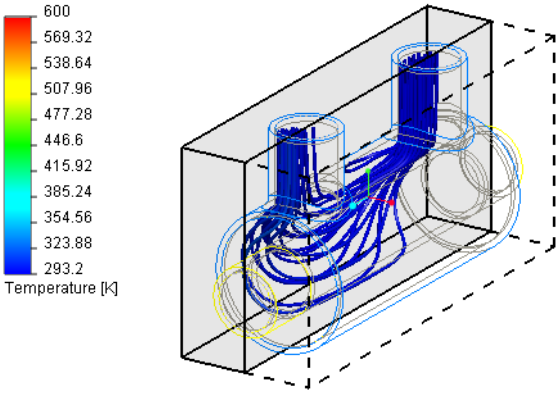


Since we are more interested in the temperature distribution, let us color the trajectories with the values of temperature.

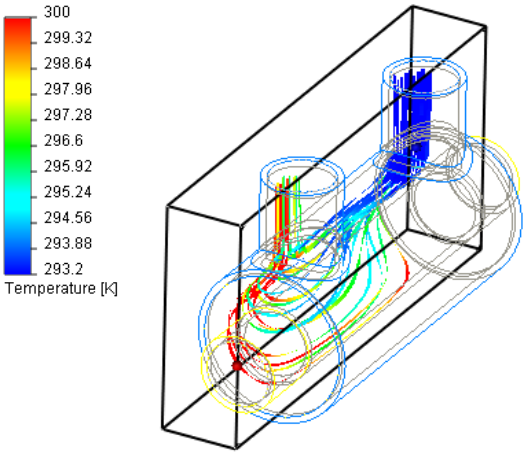
1 In the velocity palette bar click the caption with the name of the current visualization parameter and select **Temperature** in a dropdown list.



2 Click ☑. Immediately the trajectories are updated.
The water temperature range is less than the default overall (**Global**) range (293 – 600), so all of the trajectories are the same blue color. To get more information about the temperature distribution in water you can manually specify the range of interest.



Let us display temperatures in the range of **inlet-outlet** water temperature.



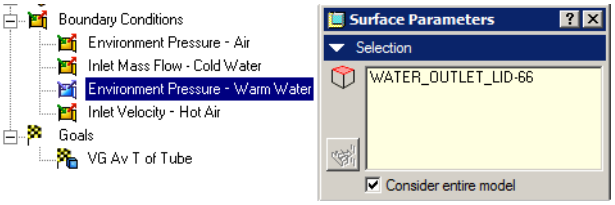
The water minimum temperature value is close to 293 K. Let us obtain the values of air and water temperatures at outlets using Surface Parameters. You will need these values to calculate the heat exchanger efficiency and determine the appropriate temperature range for flow trajectories visualization.

📖 **Surface Parameters** *allows you to display parameter values (minimum, maximum, average and integral) calculated over the specified surface. All parameters are divided into two categories: Local and Integral. For local parameters (pressure, temperature, velocity etc.) the maximum, minimum and average values are evaluated.*

Viewing the Surface Parameters

1 Click **Surface Parameters** . The **Surface Parameters** dialog appears.

2 In the FloEFD Analysis tree select the **Environment Pressure - Warm Water** item to select the inner face of the **WATER_OUTLET_LID**.

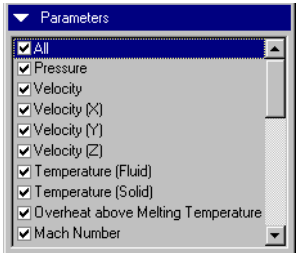


3 Select **Consider entire model** to take into account the **Symmetry** condition to see the values of parameters as if the entire model, not a half of it, was calculated. This is especially convenient for such parameters as mass and volume flow.

4 Under **Parameters**, select **All**.

5 Click **Show**. The calculated parameters values are displayed on the pane at the bottom of the screen. Local parameters are displayed at the left side of the bottom pane, while integral parameters are displayed at the right side.

6 Take a look at the local parameters.



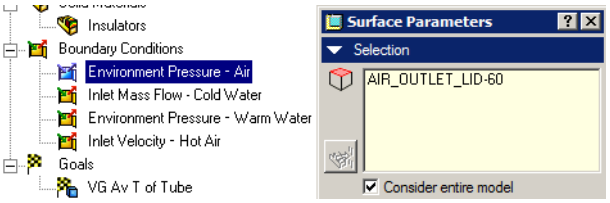
Local Parameter	Minimum	Maximum	Average	Bulk Average	Surface Area [m^2]
Pressure [Pa]	202650	202650	202650	202650	0.0039
Density [kg/m^3]	991.3	997.3	995.7	995.5	0.0039
Velocity [m/s]	2.1e-004	3.9e-003	2.6e-003	3.1e-003	0.0039
Velocity (X) [m/s]	-1.5e-004	2.1e-004	-2.4e-006	-2.4e-005	0.0039
Velocity (Y) [m/s]	1.9e-004	3.9e-003	2.6e-003	3.1e-003	0.0039
Velocity (Z) [m/s]	-1.9e-004	1.5e-004	6.7e-006	2.1e-005	0.0039
Mach Number [ ]	0	0	0	0	0.0039
Temperature (Fluid) [K]	294.4	313.5	299.8	300.4	0.0039

You can see that the average water temperature at the outlet is about 300 K.

Now let us determine the temperature of air at the outlet.

7 Click the **Environment Pressure - Air** item to select the inner face of the **AIR_OUTLET_LID**.

8 At the bottom pane, click **Refresh** .



9 Look at the local parameters at the left side of the bottom pane.

Local Parameter	Minimum	Maximum	Average	Bulk Average	Surface Area [m^2]
Pressure [Pa]	101325	101325	101325	101325	0.0039
Density [kg/m^3]	0.6	0.7	0.6	0.6	0.0039
Velocity [m/s]	8.1	10.2	9.8	9.8	0.0039
Velocity (X) [m/s]	-9.5e-002	0.1	-2.1e-002	-2.0e-002	0.0039
Velocity (Y) [m/s]	-8.7e-002	0.1	-2.6e-004	-2.6e-004	0.0039
Velocity (Z) [m/s]	-10.2	-8.1	-9.8	-9.8	0.0039
Mach Number [ ]	0.02	0.02	0.02	0.02	0.0039
Temperature (Fluid) [K]	521.1	600.0	585.5	586.0	0.0039

You can see that the average air temperature at the outlet is about 585 K.

10 The values of integral parameters are displayed at the right side of the bottom pane. You can see that the mass flow rate of air is 0.046 kg/s. This value is calculated with the **Consider entire model** option selected, i.e. taking into account the **Symmetry** condition.

Integral Parameter	Value	X-component	Y-component	Z-component	Surface Area [m^2]
Mass Flow Rate [kg/s]	-0.046				0.0078
Volume Flow Rate [m^3/s]	-0.0764				0.0078
Surface Area [m^2]	0.0078	0	0	0.0078	0.0078
Total Enthalpy Rate [W]	-27495.7				0.0078
Uniformity Index [ ]	1.9552330				0.0078
CAD Fluid Area [m^2]	0.0079				0.0079
CAD Solid Area [m^2]	0.0079				0.0079

11 Click **OK** ✔ to close the dialog box.

Calculating the Heat Exchanger Efficiency

The heat exchanger efficiency can be easily calculated, but first we must determine the fluid with the minimum capacity rate ($C=\dot{m}c$). In this example the water mass flow rate is 0.02 kg/s and the air mass flow rate is 0.046 kg/s. The specific heat of water at the temperature of 300 K is about five times greater than that of air at the temperature of 586 K. Thus, the air capacity rate is less than the water capacity rate. Therefore, according to *Ref.2*, the heat exchanger efficiency is calculated as follows:

$$\varepsilon = \frac{T_{hot}^{inlet} - T_{hot}^{outlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}},$$

where T_{hot}^{inlet} is the temperature of the air at the inlet, T_{hot}^{outlet} is the temperature of the air at the outlet and T_{cold}^{inlet} is the temperature of the water at the inlet.

We already know the air temperature at the inlet (600 K) and the water temperature at the inlet (293.2 K), so using the obtained values of water and air temperatures at outlets, we can calculate the heat exchanger efficiency:

$$\varepsilon = \frac{T_{hot}^{inlet} - T_{hot}^{outlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}} = \frac{600 - 586}{600 - 293{,}2} = 0{,}045$$

As you can see, FloEFD is a powerful tool for heat-exchanger design calculations.

Ref. 2 J.P. Holman. “Heat Transfer” Eighth edition.

B4

Mesh Optimization

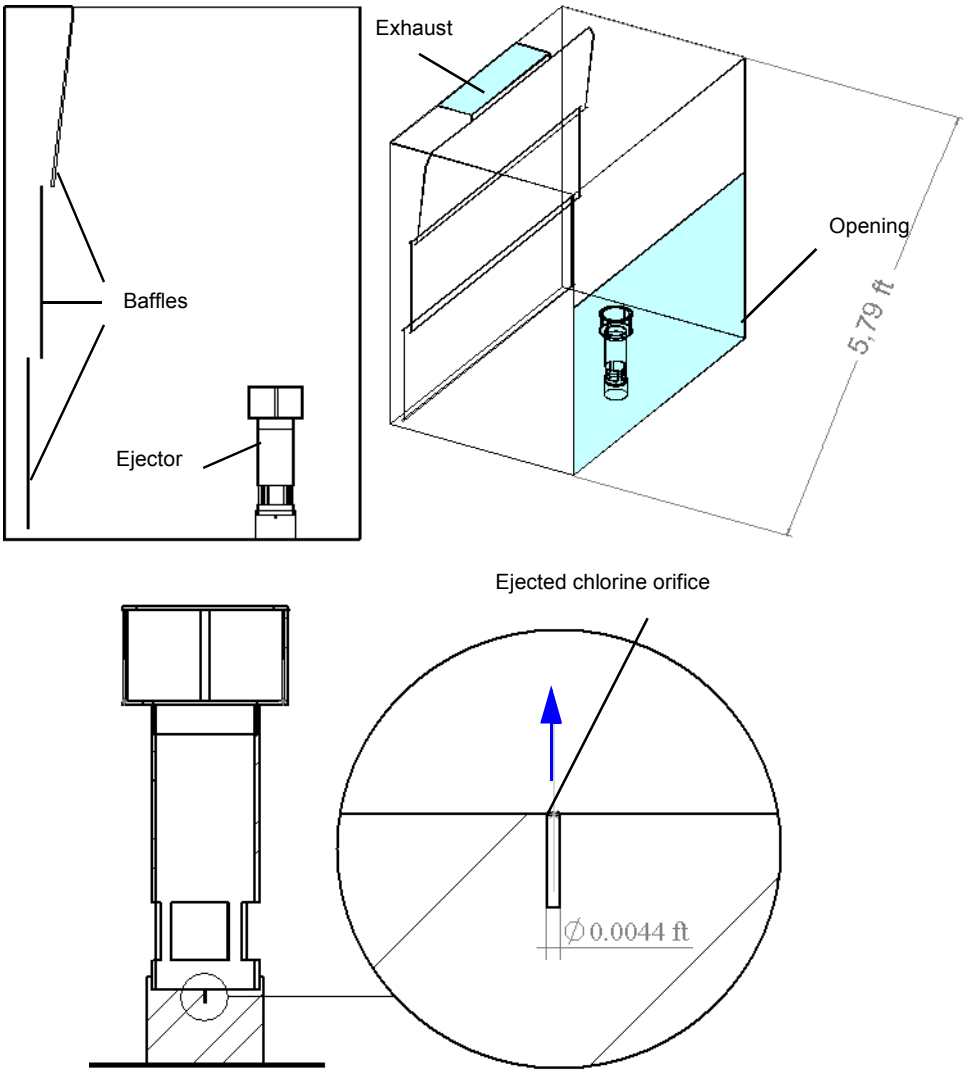
The goal of this tutorial example is to demonstrate various meshing capabilities of FloEFD allowing you to perform manual adjustment of the computational mesh. Although the automatically generated mesh is usually appropriate, intricate problems with thin and/or small, but important, geometrical and physical features can result in extremely high number of cells, for which the computer memory may be too small. In such cases we recommend you trying the FloEFD options allowing you to manually adjust the computational mesh to the solved problem's features to resolve them better. This tutorial teaches you how to do this.

The considered **Ejector in Exhaust Hood** example aims to:

- Settle the large aspect ratio between the minimum gap size and the model size by adjusting the initial mesh manually.
- Resolve small features by specifying local mesh settings.

Problem Statement

The ejector model is shown in the picture below. Note that the ejector orifice's diameter is more than 1000 times smaller than the characteristic model size determined as the computational domain's overall dimension.



Opening the Model

Copy the **B4 – Mesh Optimization** folder into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files. Open the **Ejector_in_Exhaust_ Hood.asm** assembly.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **Ejector_in_Exhaust_ Hood.asm** assembly located in the **B4 – Mesh Optimization\Ready To Run** folder and run the desired projects.*

Creating a FloEFD Project

Using the Wizard create a new project as follows:

Project name	*Ejector Analysis*
Configuration	*Use current*
Unit system	*USA*
Analysis type	*Internal; Exclude cavities without flow conditions*
Physical features	*Gravity; Default gravity (Y component: -32.1850394 ft/s^2)*
Fluids substances	*Air, Chlorine*
Wall Conditions	*Adiabatic wall, default smooth walls*
Initial Conditions	Initial gas concentration: *Air – 1, Chlorine - 0*
Result and Geometry Resolution	*Default result resolution level 3;* *Default geometry resolution: automatic minimum gap size and minimum wall thickness, other options by default*

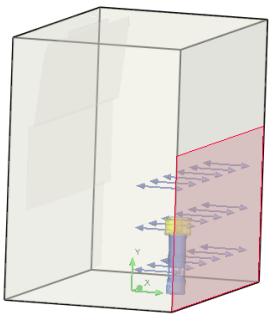
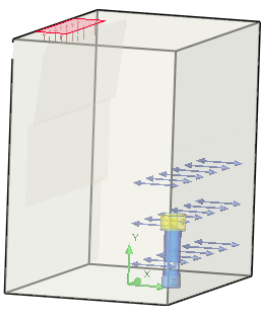
📖 *When you enable gravitation, pay attention that the hydrostatic pressure is calculated with respect to the global coordinate system, as follows:*
$P_{hydrostatic} = \rho(g_x*x + g_y*y + g_z*z)$, *where* ρ *– reference density,* g_i *- component of the gravitational acceleration vector and x, y, z - coordinates in the global coordinate system.*

Specifying Boundary Conditions

At first, let us specify all the necessary boundary conditions because they influence the automatic initial mesh settings through the automatic minimum gap size, which depends on the characteristic size of the faces on which the boundary conditions are set.

📖 *FloEFD calculates the default minimum gap size using information about the faces where boundary conditions (as well as sources, fans) and goals are specified. Thus, it is recommended to set all conditions before you start to analyze the mesh.*

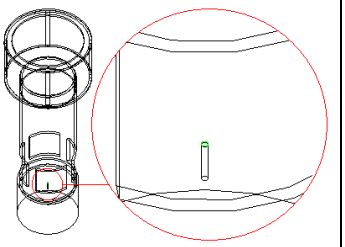
The first two boundary conditions are imposed on the exhaust hood's inlet and outlet.

Inlet Boundary Condition	***Environment Pressure:*** *Default values (14.6959 lbf/in^2, gas substance – Air) of the **Environment pressure** and **Temperature** (68.09 °F) at the lid that closes the opening (make sure that you have selected the inner face of the lid);*	
Outlet Boundary Condition	***Outlet Volume Flow**: Outlet volume flow rate of 1000 ft^3/min at the lid that closes the exhaust (make sure that you have selected the inner face of the lid).*	

Open the **Initial Mesh** dialog box (click **Flow Analysis > Initial Mesh**) and select the **Manual specification of the minimum gap size** option. You will see that the current automatic minimum gap size is 0.5 ft, which is the width of the outlet opening. Click **Cancel** to close this dialog box.

✎ *To open **Initial Mesh** dialog box in Creo Parametric, click **Flow Analysis > Mesh > Initial Mesh**.*

The next inlet volume flow rate condition defines the gas ejected from the bottom of the **Ejector** component.

Inlet Boundary Condition	***Inlet Volume Flow:*** *Inlet chlorine (**Substance concentrations**: Chlorine – 1; Air – 0) volume flow rate of 0.14 ft^3/min at the lid that closes the orifice (make sure that you have selected the upper face of the lid).*	

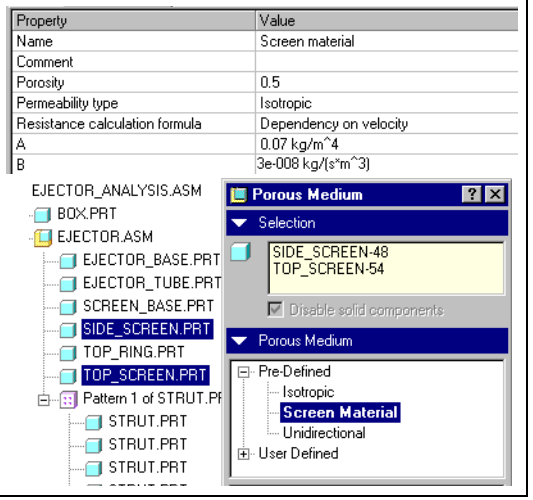
If you now look at the automatic minimum gap size value (click **Flow Analysis > Initial Mesh > Manual specification of the minimum gap size**), you will notice that it is now changed to approximately 0.00446 ft, which is close to the orifice diameter.

📖 *The Minimum gap size is a parameter governing the computational mesh, so that a certain number of cells per the specified gap should be generated. To satisfy this condition the corresponding parameters governing the mesh are set by FloEFD (number of basic mesh cells, small solid features refinement level, narrow channel resolution, etc.). Note that these parameters are applied to the whole computational domain, resolving all its features of the same geometric characteristics (not only to a specific gap).*

Since the minimum gap size value influences the mesh in the entire computational domain, the large aspect ratio between the model and the minimum gap size value will produce a non-optimal mesh: not only will all small gaps be resolved, but there will also be many small cells in places where they are not necessary. As a result, an extremely large mesh will be produced, which may result in overly large computer memory requirements exceeding the computers' available resources. Moreover, if the aspect ratio between the model and the minimum gap size is more than 1000, FloEFD may not adequately resolve such models with the automatically generated mesh anyway.

Finally, let us create the ejector’s porous media and apply it to the ejector’s top and side screens.

The material you are going to create is already defined in the Engineering Database under the Pre-Defined folder. You can skip the definition of the porous material, then when creating the porous condition, select the pre-defined "Screen Material" from the Engineering database.

Porous Media	***Screen material:*** *Porosity: 0.5,* *Permeability type: Isotropic,* *Dependency on velocity:* *A = 0.07 kg/m^4,* *B = 3e-008 kg/(s*m^3).* ***Components to apply:*** *TOP_SCREEN* *SIDE_SCREEN*	

To see advantages of using the local mesh and refinement options, let us first try to generate the computational mesh governed by the automatic mesh settings. The resulting mesh will consist of more than 1000000 cells, and may be not processed by some computers due to the computer memory restriction (you may get a warning message about insufficient memory)

Manual Specification of the Minimum Gap Size

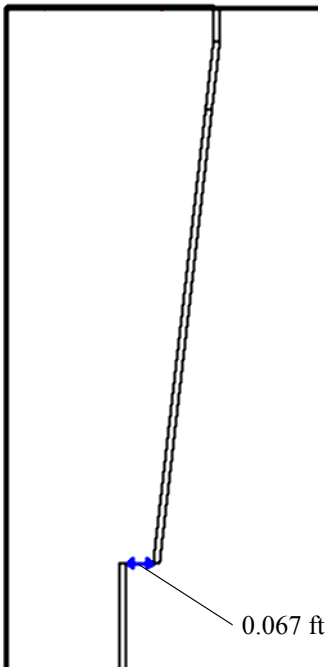
We can distinguish two parts of the model that are very different in size: a relatively big cavity having several thin walls within and no small solid features, and the ejector's region containing some very fine geometrical features. Therefore, the mesh required to resolve the ejector properly and the mesh appropriate for the rest of the model should be also very different in terms of cell size. Since the ejector region is a part of the entire computational domain, we need to specify such settings for the automatic mesh generation that the model's geometry outside the ejector's region will be resolved without excessive mesh splitting.

The minimum gap size value, automatically defined from the dimensions of the ejector's **Top Screen** and **Side Screen** components, is too small and will result in excessive mesh splitting.

To define an appropriate minimum gap size we need to examine all narrow flow passages outside the ejector's region:

- Boundary conditions;
- The passages connecting the ejector's internal volume with the model's cavity;
- The narrow flow passages between the baffles.

After reviewing the model we can accept the width of the gap between the middle and upper baffles as the minimum gap size. To avoid excessive mesh splitting, we will specify the same value for the minimum wall thickness.



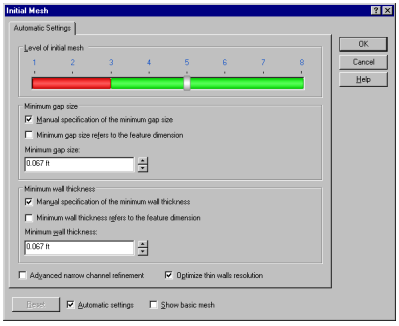
1 Click **Flow Analysis > Initial Mesh**.

*In Creo Parametric, click **Flow Analysis > Mesh > Initial Mesh**.*

2 Use the slider to set the **Level of the initial mesh** to **5**.

3 Select the **Manual specification of the minimum gap size** checkbox and enter `0.067 ft` in the **Minimum gap size** box.

4 Select the **Manual specification of the minimum wall thickness** checkbox and enter `0.067 ft` in the **Minimum wall thickness** box.



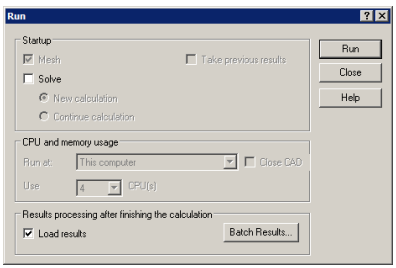
5 Click **OK**.

To view the resulting mesh:

1 Click **Flow Analysis > Run**

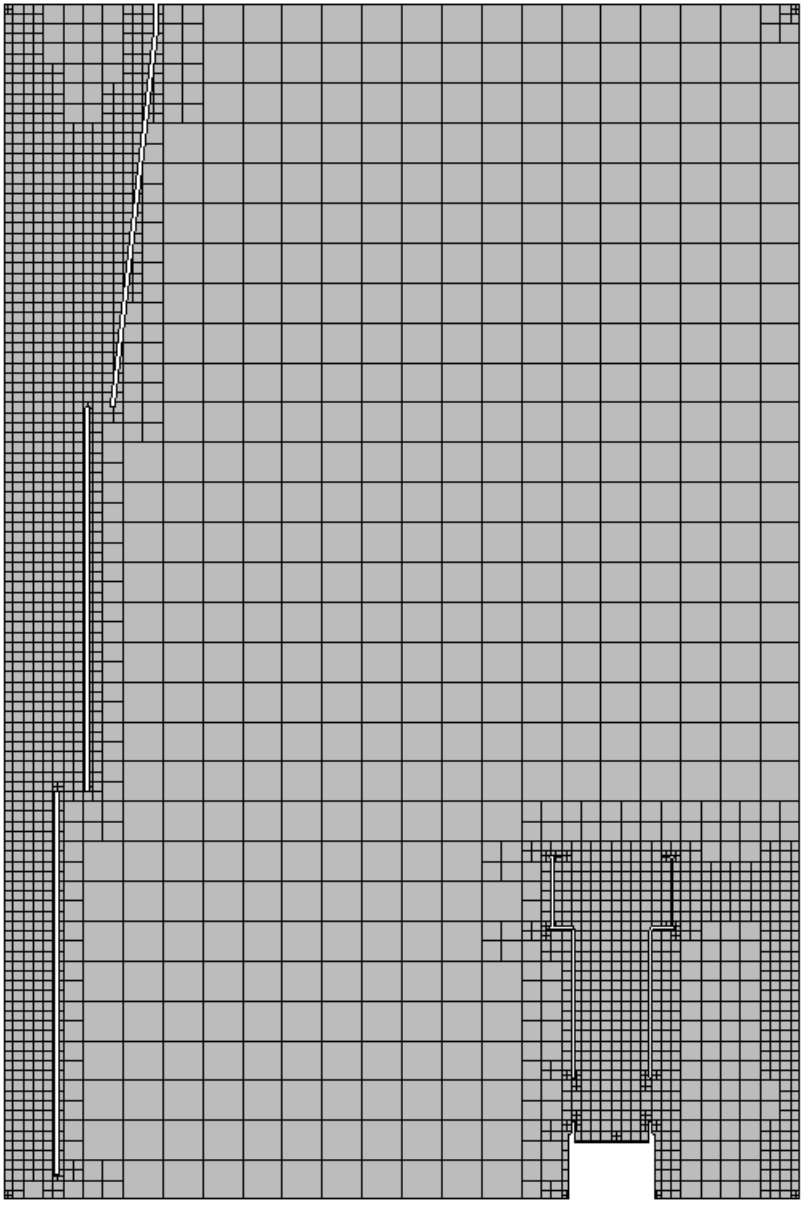
2 Clear the **Solve** check box in order to generate the mesh only.

3 Click **Run**.



After the mesh generation finishes you can view it by creating a **Cut Plot** on the ASM_RIGHT plane with the **Mesh** option selected.

The resulting mesh has significantly less cells than the mesh generated automatically with the default values of **Minimum gap size** and **Minimum wall thickness**. The total number of cells is about 200 000.



Switching off the Automatic Mesh Definition

We have successfully reduced the number of cells, yet using the mesh of the higher level. The higher level mesh provides better refinement in the regions with small geometrical features. However, we actually do not need such a fine mesh in some regions where the flow field changes slowly and so does not affect the solution much. We can further decrease the number of cells by switching off the automatic definition of the mesh generation settings and adjusting these settings manually. The decreased number of cells will provide us a computer memory reserve needed to resolve better fine geometrical features of the ejector.

Click **Flow Analysis > Project > Rebuild**.

1 Click **Flow Analysis > Initial Mesh**. Switch off the automatic mesh settings by clearing the **Automatic settings** check box. The **Initial Mesh** dialog box controls the basic mesh and the initial mesh within the entire computational domain unless local initial mesh settings are specified.

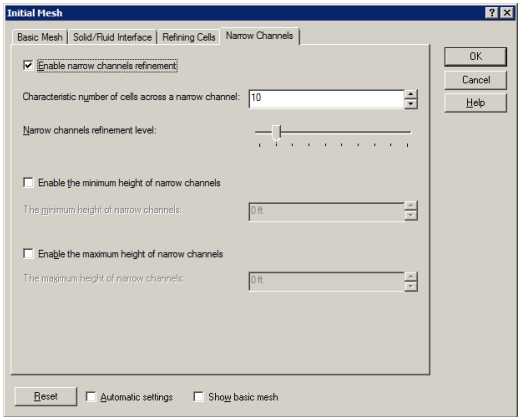
✎ *To open **Initial Mesh** dialog box in Creo Parametric, click **Flow Analysis > Mesh > Initial Mesh**.*

🕮 *The mesh is named **Initial** since it is the mesh the calculation starts from and it could be further refined during the calculation if the solution-adaptive meshing is enabled. The initial mesh is constructed from the **Basic** mesh by refining the basic mesh cells in accordance with the specified mesh settings. The Basic mesh is formed by dividing the computational domain into slices by parallel planes which are orthogonal to the Global Coordinate System's axes.*

The Initial Mesh's parameters are currently set by FloEFD in accordance with the previously specified automatic mesh settings, including **Minimum gap size** and **Minimum wall thickness**.

1 Go to the **Narrow channel** tab and set the **Narrow channels refinement level** to 1. This allows us to reduce the number of cells in the channels between the baffles and the wall of the **Box**.

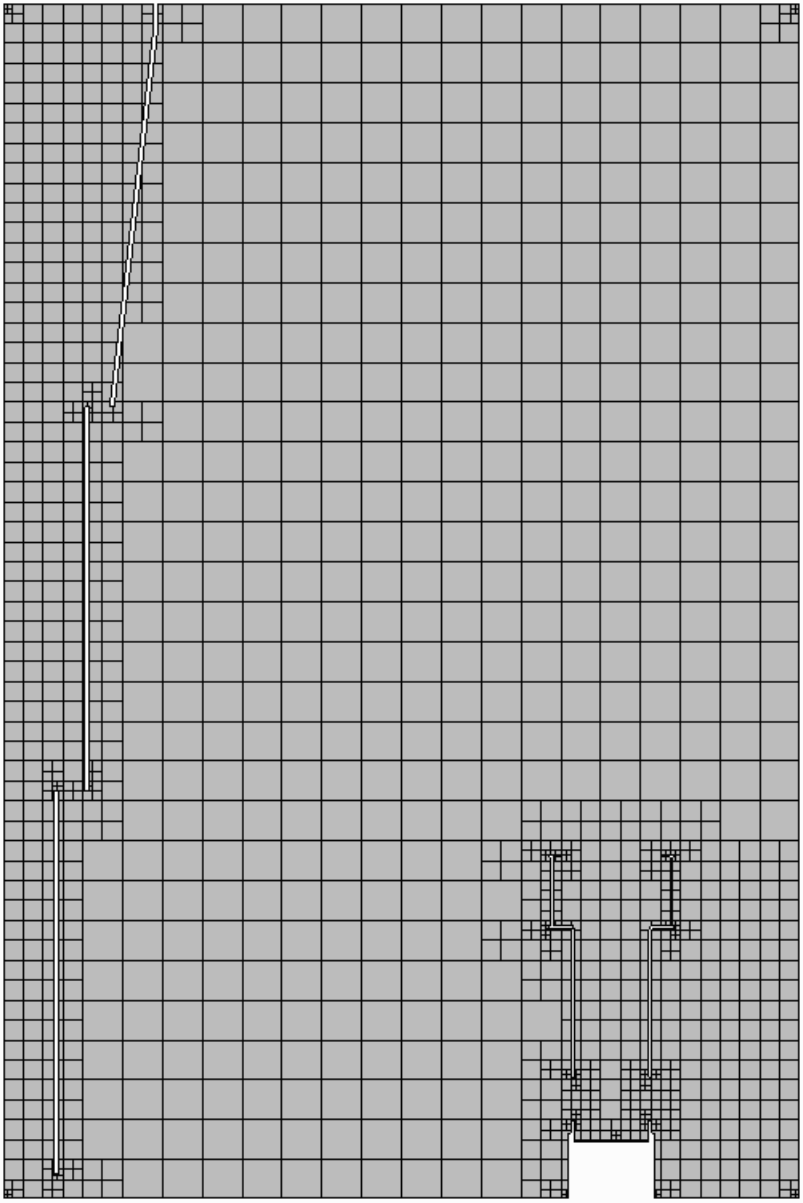
🕮 *The **Narrow channels refinement level** specifies the smallest size of the cells in model's flow passages with respect to the basic mesh cells.*



*So if N = 0…9 is the specified **Narrow channels refinement level**, the minimum size of the cells obtained due to the mesh refinement is 2^N times smaller (in each direction of the Global Coordinate System, or 8^N times by volume) than the basic mesh cell's size.*

To view the resulting mesh, run the mesh generation again (without further problem calculation).

The resulting mesh is shown below. It consists of about 80 000 cells.



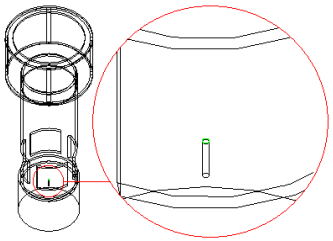
Using the Local Initial Mesh Option

While the ejector's geometry is resolved reasonably well, the inlet face of the ejector's orifice needs finer mesh in order to resolver it properly. The resolution of the boundary condition face is very important for imposing the boundary condition correctly. To resolve the gas inlet face properly we will use the **Local Initial Mesh** option.

The local initial mesh option allows you to specify mesh settings in a local region of the computational domain to resolve better the model geometry and/or flow peculiarities in this region. The local region can be defined by a component of the assembly (disabled in the **Component Control** dialog box, in case it belongs to the fluid region) or specified by selecting a face, edge or vertex of the model. Local mesh settings are applied to all cells intersected by a component, face, edge, or a cell enclosing the selected vertex.

1 Click **Flow Analysis > Insert > Local Initial Mesh**.

*In Creo Parametric click **Flow Analysis > Mesh > Local Initial Mesh**.*

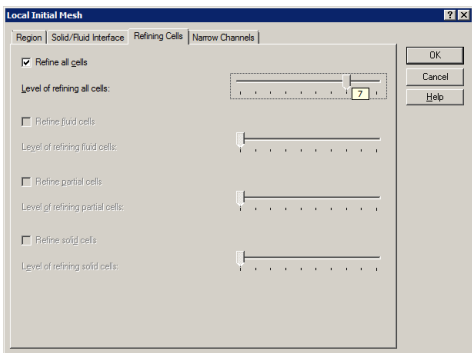


2 Select the inlet face of the ejector's orifice or click the **Inlet Volume Flow 1** boundary condition in the FloEFD Analysis tree to select the face on which this boundary condition is applied.

3 Clear the **Automatic settings** check box and switch to the **Refining cells** tab.

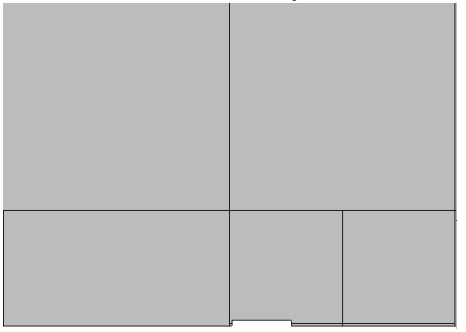
4 Select the **Refine all cells** check box and use the slider to set the **Level of refining all cells** to 7.

5 Click **OK**.

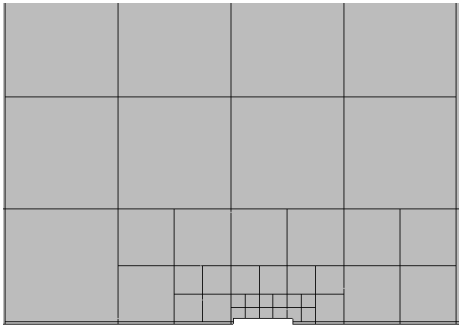


To see the resulting mesh, once again run the mesh generation.

Now we have specified to refine all cells near the ejector's orifice inlet face up to the maximum level. The locally refined mesh is shown below.



Before specifying local initial mesh



After specifying local initial mesh

Specifying Control Planes

The basic mesh in many respects governs the generated computational mesh. The proper basic mesh is necessary for the most optimal mesh.

You can control the basic mesh in several ways:

- Change number of the basic mesh cells along the X, Y, Z-axes.
- Shift or insert basic mesh planes.
- Stretch or contract the basic mesh cells locally by changing the relative distance between the basic mesh planes.

🕮 *The local mesh settings do not influence the basic mesh but are basic mesh sensitive: all refinement levels are set with respect to the basic mesh cell.*

You may notice that the mesh resolving the ejector's orifice inlet face is not symmetric. It can has a negative effect on the specified boundary condition. We will add a control plane to shift the boundary between cells so that it will pass through the center of the inlet face.

1 Click **Wireframe** button on the **Model Display** toolbar. The wireframe display mode makes it easier to select small geometrical features.

✐ *In Creo Parametric click* ***Model > Display Style > Wireframe*** *on the* ***Graphics Toolbar****.*

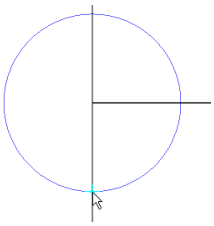
2 In the **Initial Mesh** dialog box, go to the **Basic Mesh** tab.

3 Click **Add Plane**. The **Create Control Planes** dialog box appears.

4 In the **Creating mode** list select **Reference geometry**.

5 Under **Parallel to** select **XY**.

6 Zoom in to the ejector's orifice area and select one of the vertexes of the the edge of the inlet face in the graphics area. The control plane will pass through the middle of the edge parallel to the Global Coordinate System plane selected in the **Parallel to** group.



Please check that the value of offset along the Z axis, appeared in the **Control planes** list, is equal to 0.703125 ft. If not, it means that you have mistakenly selected another geometry feature. In this case, right-click on the **Control planes** list and select **Delete All**, then try to select the edge of the inlet face again.

7 Click **OK**. The **Z2** control plane appears in the **Control intervals** table.

📖 *You can visualize the basic mesh before solving the problem. To see the basic mesh, click **Show basic mesh** in the **Initial Mesh** dialog box or click **Flow Analysis > Project > Show Basic Mesh**.*

8 Click **OK** to save changes and close the **Initial Mesh** dialog box.

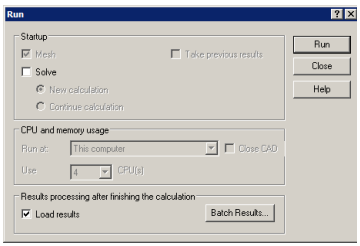
Then, generate the initial mesh to check whether the thin walls and the other geometry are resolved.

1 Click **Flow Analysis > Solve > Run**.

2 Clear the **Solve** check box in order to generate the mesh only.

3 Clear the **Load results** check box.

4 Click **Run**.



Prior to visualizing the initial computational mesh, let us switch the FloEFD option to use the meshed geometry instead of the model's geometry to visualize the results.

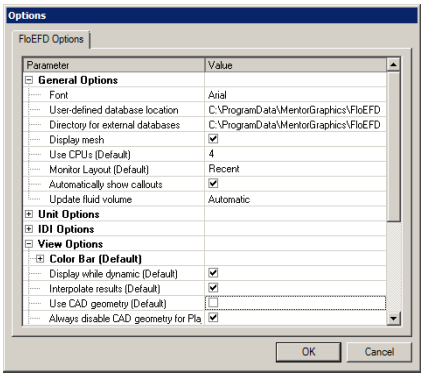
By default, FloEFD shows the model's geometry when displaying the results. Depending on how exactly the model has been resolved with the computational mesh, the model's geometry may differ from the geometry used in the calculation. To display the real captured geometry the **Use CAD geometry** option is reserved.

1 Click **Flow Analysis > Tools > Options**.

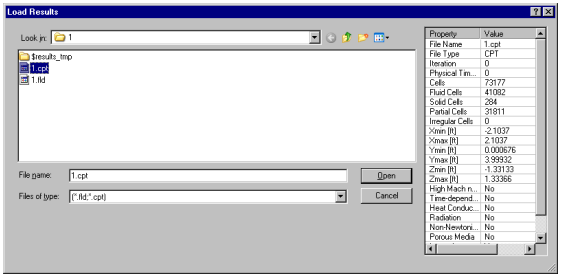
2 On the **FloEFD Options** tab, under **General Options**, select the **Display mesh** check box.

3 Under **View Options** clear the **Use CAD geometry (Default)** check box.

4 Click **OK**.



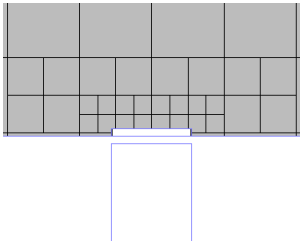
Next load the file with the initial computational mesh: right-click the **Results** icon and select **Load Results**, then select the **1.cpt** file and click **Open**. Note that the total number of cells is about 75 000.

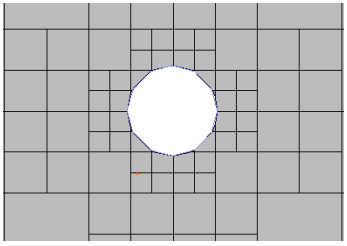


The calculation results, including the current computational mesh, are saved in the .fld files, whereas the initial computational mesh is saved separately in the .cpt files. Both of the files are saved in the project folder, whose numerical name is formed by FloEFD and must not be changed.

Create a cut plot based on the ASM_RIGHT planewith the **Mesh** option selected. Create a second cut plot based on the ejector's orifice inlet face with the **Position** of -0.00015 ft relative to the selected face and the same settings as the first cut plot.

Now you can see that the generated mesh is symmetrical relative to the center of the inlet face.





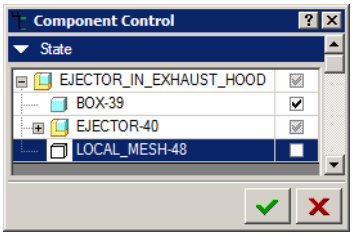
Creating a Second Local Initial Mesh

With the specified mesh settings the ejector's geometry will be resolved properly. But we need to create the mesh successfully resolving not only fine geometrical features, but the small flow peculiarities as well. In the **Ejector Analysis** project such peculiarities can be found within the internal volume of the ejector, where the thin stream of chlorine is injected from the ejector's orifice. Therefore the mesh within the ejector's region must be split additionally. To refine the mesh only in this region and avoid excessive splitting of the mesh cells in other parts of the model, we apply a local initial mesh at the component surrounding this region. The component was created specially to specify the local initial mesh.

Set to resolved the suppressed **LOCAL_MESH** component. Click **Close** after FloEFD shows you a warning message. Note that this component was created so that there is a small distance between the boundaries of the component and the solid feature of interest (i.e., the ejector). Because the local settings are applied only to the cells whose centers lie within the selected model component, it is recommended to have the component's boundaries offset from the solid component's walls.

After resolving the **LOCAL_MESH** component an error message appears informing you that the inlet volume flow condition is not in contact with the fluid region. The problem disappears after disabling the component in the **Component Control** dialog box to treat it as a fluid region.

Click **Flow Analysis > Component Control** and deselect the **LOCAL_MESH-48** component. Click **OK**.



Rebuild the project by clicking **Flow Analysis >Project > Rebuild**.

🕮 *You can also disable components directly from the* **Local Initial Mesh** *dialog box by selecting the* **Disable solid components** *option on the* **Region** *tab.*

Next specify the local mesh settings for the ejector's region.

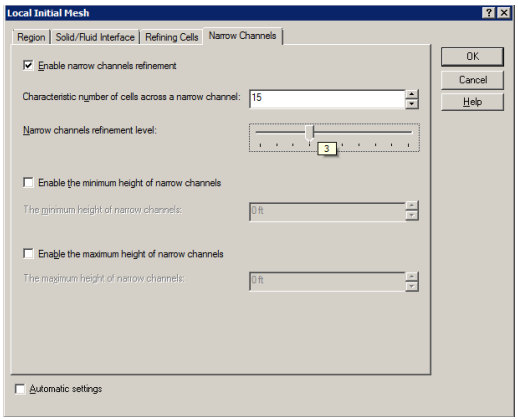
1 Select the **LOCAL_MESH** component.

2 Click **Flow Analysis > Insert > Local Initial Mesh**.

*In Creo Parametric click **Flow Analysis > Mesh > Local Initial Mesh**.*

3 Clear the **Automatic settings** check box and switch to the **Narrow Channels** tab.

4 Specify the **Characteristic number of cells across a narrow channel** equal to `15`.



5 Use the slider to set the **Narrow channels refinement level** to `3`.

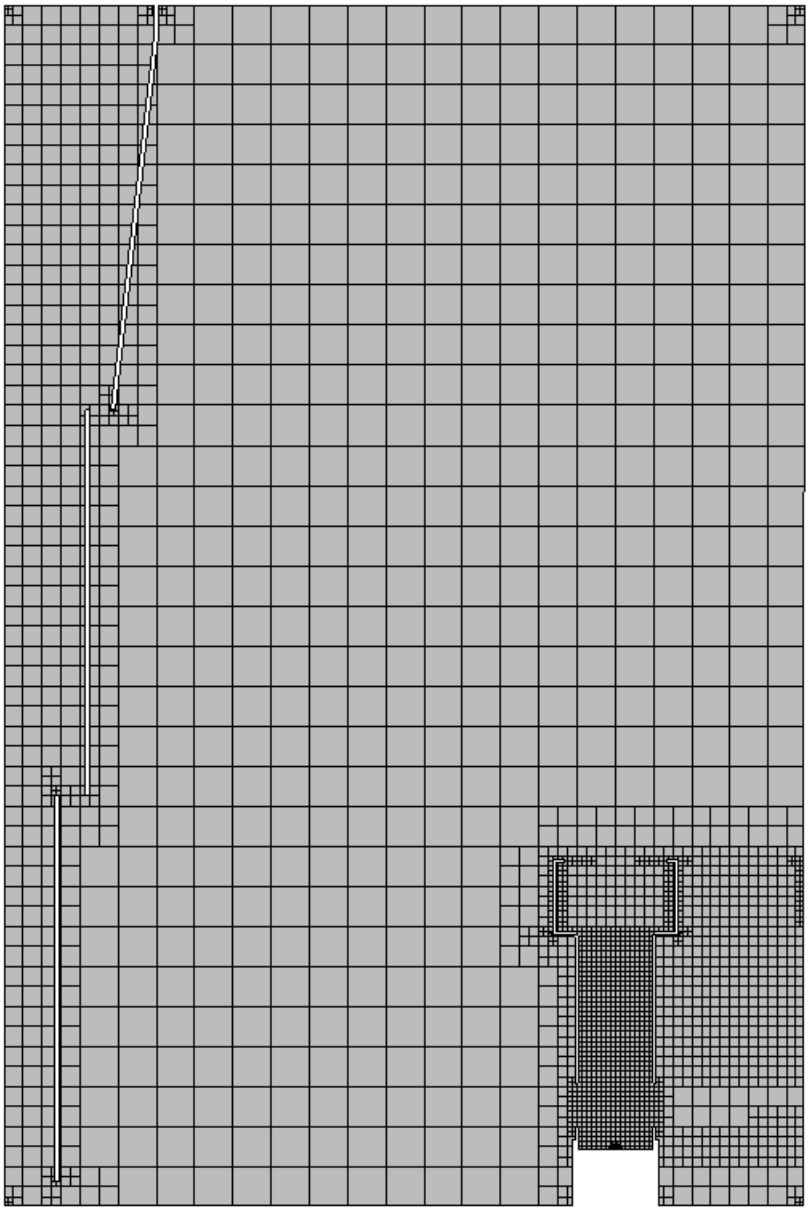
6 Click **OK**.

*The settings on the **Narrow Channels** tab controls the mesh refinement in the model's flow passages. **Characteristic number of cells across a narrow channel** box specify the number of initial mesh cells (including partial cells) that FloEFD will try to set across the model's flow passages in the direction normal to solid/fluid interface . If possible, the number of cells across narrow channels will be equal to the specified characteristic number, otherwise it will be close to the characteristic number. If this condition is not satisfied, the cells lying in this direction will be split to satisfy the condition.*

Rebuild the project and run the mesh generation again.

Click **Flow Analysis > Results > Display > Geometry**, then select `EJECTOR.ASM and LOCAL_MESH.PRT` in the model tree, right click and press **hide** to hide the model.

In the figure below you can see the final mesh. After all the adjustments made to resolve only the regions of interest its number of cells turned out to be about 100 000. The is an order less comparing to the mesh generated using the automatic mesh settings, where the number of cells turned out to be more than 1 000 000.



C

Advanced Examples

The **Advanced Examples** presented below demonstrate how to use a wide variety of the FloEFD features to solve real-life engineering problems. It is assumed that you successfully completed all First Steps examples before.

C1 - Application of EFD Zooming

C2 - Textile Machine

C3 - Non-Newtonian Flow in a Channel with Cylinders

C4 - Radiative Heat Transfer

C5 - Rotating Impeller

C6 - CPU Cooler

C7 - Oil Catch Can

Advanced Examples:

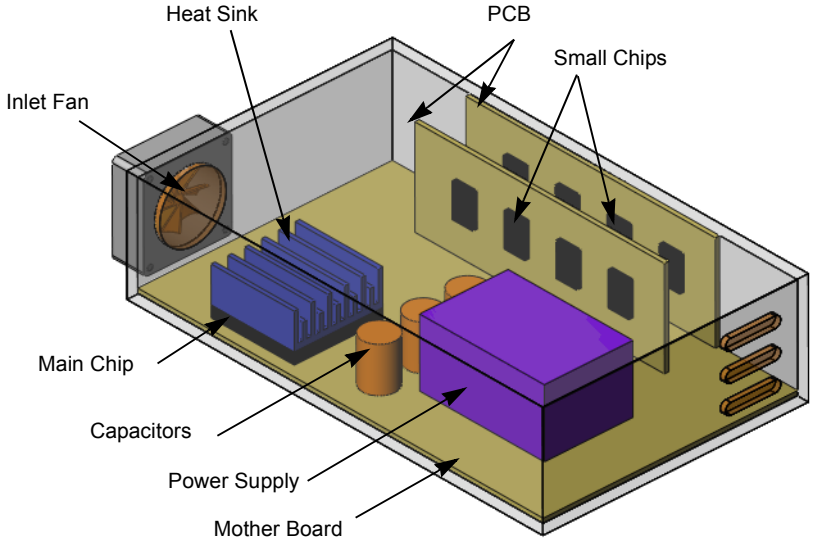
C1

Application of EFD Zooming

Problem Statement

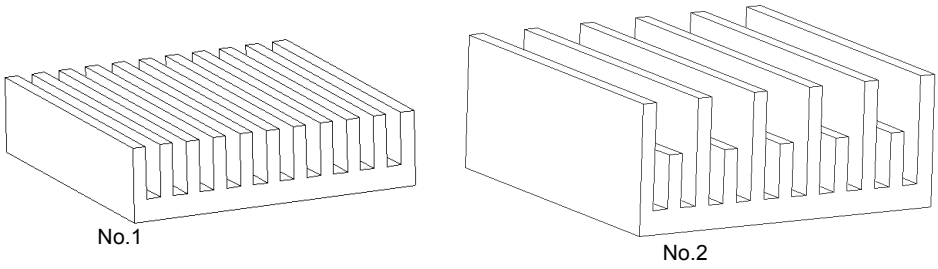
The FloEFD capability of EFD Zooming is demonstrated as an engineering tutorial example of selecting a better heat sink shape for a main chip taking into account other electronic components in an electronic enclosure.

The assembly model of the electronic enclosure including the main chip's heat sink under consideration is shown in the figure below. The fan installed at the enclosure inlet blows air through the enclosure to the outlet slots with the goal of cooling the heated electronic elements (having heat sources inside). The planar main chip is attached to a motherboard made of an insulator. To cool the main chip better, its opposite plane surface is covered by a heat sink cooled by the air stream from the fan.



Electronic enclosure

The problem's engineering aim is to determine the temperature of the main chip when using one of two heat sink designs considered with the other conditions unchanged within the enclosure. As a result, we will determinate difference in cooling capability between these two competing shapes.



The heat sink's competing shapes (No.1 and No.2)

As you can see, all components within the electronic enclosure except the main chip's heat sink are specified as coarse shapes without any small details, since they do not influence the main chip's temperature which is the aim of the analysis (the enclosure model was preliminary simplified to this level on purpose). On the contrary, the heat sink of each shape is featured by multiple thin (thickness of 0.1 in) fins with narrow (gaps of 0.1 in) channels between them.

To solve this problem, FloEFD offers two possible approaches described below.

In the first and more direct way, we compute the entire flow inside the whole electronic enclosure for each heat sink shape using the **Local Initial Mesh** option for constructing a fine computational mesh in the heat sink's narrow channels and thin fins. Naturally, the **Heat conduction in solids** option is enabled in these computations.

In the other, two-stage way (EFD Zooming using the **Transferred Boundary Condition** option), we solve the same problem in the following stages:

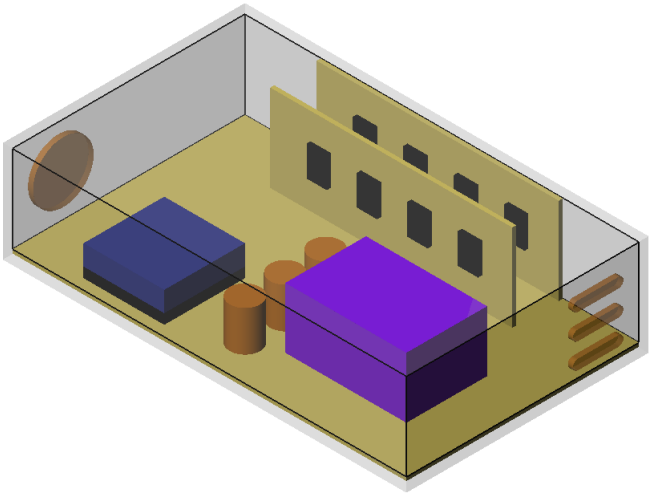
1. computing the entire flow inside the whole electronic enclosure at a low result resolution level without resolving the heat sink's fine features (so, the parallelepiped envelope is specified instead of the heat sink's comb shape) and disabling the **Heat conduction in solids** option;
2. computing the flow over the real comb-shaped heat sink in a smaller computational domain surrounding the main chip, using the **Transferred Boundary Condition** option to take the first stage's computation results as boundary conditions, specifying a fine computational mesh in the heat sink's narrow channels and thin fins to resolve them, and enabling the **Heat conduction in solids** option.
   The first stage's computation is performed once and then used for the second stage's computations performed for each of the heat sink's shapes.

The EFD Zooming Approach to Solve the Problem

Let us begin from the second (EFD Zooming) approach employing the **Transferred Boundary Condition** option. Then, to validate the results obtained with this approach, we will solve the problem in the first way by employing the **Local Initial Mesh** option.

First Stage of EFD Zooming

In accordance with the 1st stage of EFD Zooming aimed at computing the entire flow inside the electronic enclosure, it is not necessary to resolve the flow's small features, i.e. streams between the heat sink's fins, at this stage. Therefore, we suppress the heat sink's comb shape feature in the assembly model, obtaining the parallelepiped envelope instead.



A parallelepiped heat sink is used at the 1st stage of EFD Zooming.

The model simplification at this stage allows us to compute the electronic enclosure's flow by employing the automatic initial mesh settings with a lower level of initial mesh (we use 4) and accepting the automatic settings for the minimum gap size and the minimum wall thickness. Moreover, at this stage it is also not necessary to compute heat conduction in solids, since we do not compute the main chip temperature at this stage. Instead, we specify surface heat sources of the same (5W) heat transfer rates at the main chip and heat sink (parallelepiped) faces and at the small chips' faces (they are heated also in this example) to simulate heating of the air flow by the electronic enclosure. This is not obligatory, but removing the heat conduction in solids at this stage saves computer resources. As a result, the computer resources (memory and CPU time) required at this stage are substantially reduced.

Project for the First Stage of EFD Zooming

Opening the Model

Copy the **C1 - EFD Zooming** folder into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files.Open the **enclosure_assembly.asm** assembly. In the **Select Instance** dialog box, select the **GLOBAL** instance. Notice that the heat sink (**HEATSINK_NO1_SC.PRT**) has its cuts suppressed, so it looks like a parallelepiped.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **enclosure_assembly.asm** assembly located in the **C1 - EFD Zooming\Ready To Run** folder and run the desired projects.*

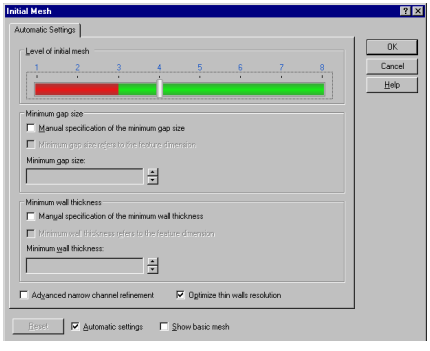
Creating a FloEFD Project

Using the **Wizard** create a new project as follows:

Project name	*Zoom – Global – L4*
Configuration name	*GLOBAL*
Unit system	*USA*
Analysis type	*Internal; Exclude cavities without flow conditions*
Physical features	*No physical features are selected*
Fluid	*Air*
Wall Conditions	*Adiabatic wall, Zero roughness*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Result resolution level set to 4, other options are default*

For this project we use the automatic initial mesh and the default computational domain.

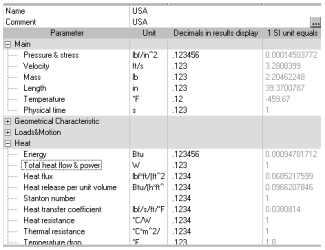
📖 *Notice that **Level of initial mesh** is set to 4 in accordance with the **Result resolution level** specified in the Wizard. The **Result Resolution** defines two parameters in the created project, namely, the **Level of initial mesh** and the **Results resolution level**. The **Level of initial mesh** is accessible from the **Initial Mesh** dialog box and governs the initial mesh only. The **Results resolution level** is accessible from the **Calculation Control***

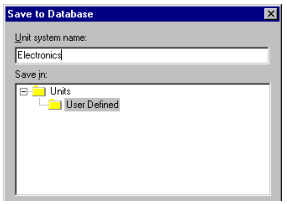


Options *dialog box and governs the refinement of computational mesh during calculation and the calculation finishing conditions. The* ***Geometry Resolution*** *options, which also influence the initial mesh, can be changed in the* ***Initial Mesh*** *box, and/or their effects can be corrected in the* ***Local Initial Mesh*** *dialog box.*

Specifying Unit System

After passing the Wizard, first we will adjust the system of units.





1 Click **Flow Analysis > Units**.

In Creo Parametric click ***Flow Analysis > Project > Units***.

2 Specify **Inch** for the **Length** and **Watt** for the **Total Heat flow & power**.

3 Click **Save**.

4 In the **Save to Database** dialog box, expand the **Units** group and select the **User Defined** item.

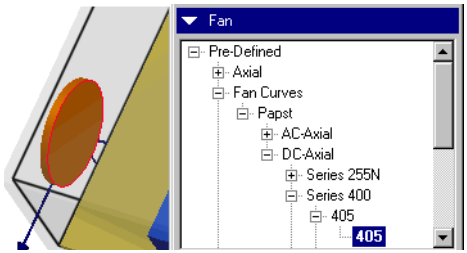
5 Name the new system of units `Electronics`.

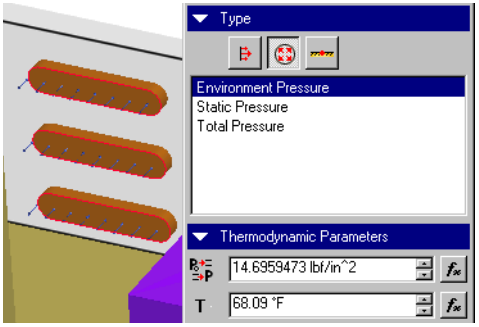
6 Click **OK** to return to the **Unit System** dialog box.

7 Click **OK**.

Specifying Boundary Conditions

We will specify External Inlet Fan at the inlet, Environment Pressure at three outlets. For more detailed explanation of how to set these conditions, you can refer to the **Conjugate Heat Transfer** tutorial.

Inlet Boundary Condition	***External Inlet Fan****: Pre-Defined \Fan Curves\ PAPST\ DC-Axial\ Series 400\ 405\ 405 with default settings (environment pressure of 14.6959 lbf/in^2, temperature of 68.09 °F) set at the* ***INLET_LID****;*	

Outlet Boundary Condition	***Environment Pressure**: Default thermodynamic parameters (ambient pressure of 14.6959 lbf/in^2, temperature of 68.09 °F) for the Environment pressure at the **OUTLET_LID**s.*	

Specifying Heat Sources

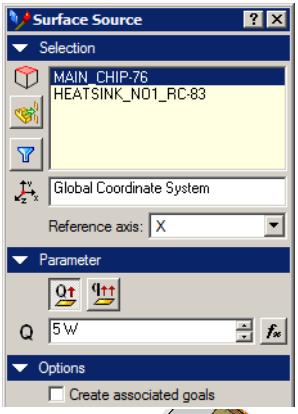
As mentioned earlier in this chapter, to simulate the flow heating by the electronic enclosure, we specify surface heat sources of the same (5W) heat transfer rates at the main chip and the heat sink (parallelepiped) faces and at the small chips' faces. Since we do not consider heat conduction in solids in this project, the surface heat source is applied only to the faces contacting with fluid.

1 Click **Flow Analysis > Insert > Surface Source**.

2 In the Model Tree, select the **HEATSINK_NO1_SC** and **MAIN_CHIP** components.

The selected components appear in the **Faces to Apply the Surface Source** list.

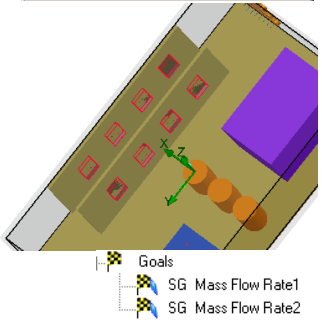
3 Under **Parameter**, set the **Heat Transfer Rate** Q to 5 W.



The specified heat source value (Heat Transfer Rate) is distributed among the selected faces in proportion to their areas. Only faces contacting with fluid are taken into account.

4 Click **OK**.

Following the same procedure, create a surface source of 5 W on the fluid-contacting surfaces of small chips.



Specifying Goals

Specify the surface goals of mass flow rate at the inlet and outlet.

Run the calculation. After the calculation is finished, you can start the second stage of EFD Zooming to focus on the main chip.

Save the model.

Second Stage of EFD Zooming

At the 2nd stage of EFD Zooming aimed at determining the main chip's temperature, we compute the flow over the heat sink in a smaller computational domain surrounding the main chip, using the **Transferred Boundary Condition** option to take the first stage's computation results as boundary conditions. To compute the solids temperature, we enable the **Heat conduction in solids** option. Since at this stage the computational domain is reduced substantially, a fine computational mesh with an affordable number of cells can be constructed in the heat sink's narrow channels and thin fins, even when considering heat conduction in solids during computation.

Project for the Second Stage of EFD Zooming

Opening the Model

Activate the **SINK_1_ZOOM** configuration (instance).

Creating a FloEFD Project

Using the **Wizard** create a new project as follows:

Project name	*Zoom – Sink 1 – L4*
Configuration name	*SINK_1_ZOOM*
Unit system	*Electronics*
Analysis type	*Internal*
Physical features	*Heat conduction in solids is enabled*
Fluid	*Air*
Default solid	*Metals/Aluminum*
Wall Condition	*Adiabatic wall, Zero roughness*
Initial Conditions	*Default initial conditions (in particular, the initial solid temperature is 68.09°F)*
Result and Geometry Resolution	*Result resolution level set to 4; Minimum gap size = 0.1 in, automatic minimum wall thickness; other options are default.*

Here, we use the automatic initial mesh by specifying the **Result resolution** level (**Level of initial mesh**) of 4, but in contrast to the first stage's computation, we specify manually the minimum gap size of 0.1 in to resolve the fine features of heat sink.

Next, we will reduce the computational domain to focus on the main chip, i.e. perform EFD Zooming.

Adjusting the Computational Domain Size

When reducing the computational domain for EFD Zooming purposes, it is necessary to take into account that the first stage's computation results will serve as the boundary conditions at this domain's boundaries. Therefore, to obtain reliable results in the second stage's computations, we have to specify computational domain boundaries (as planes parallel to the X-, Y-, Z-planes of the Global Coordinate system) satisfying the following conditions:

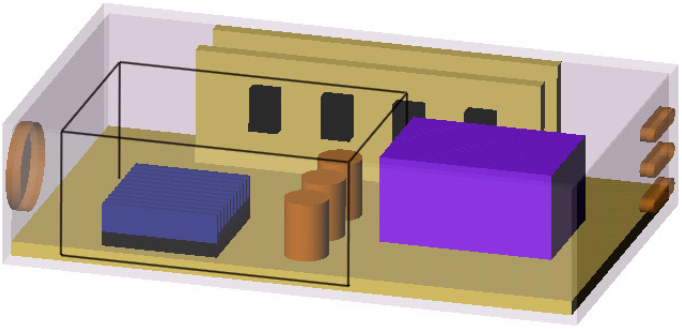
1. the flow and solid parameters at these boundaries, taken from the first stage's computation, must be as uniform as possible;
2. the boundaries must not lie too close to the object of interest, since the object's features were not resolved at the first stage's computation. The computational domain must be large enough not to receive influence from more complex features of the newly added object;
3. the boundary conditions transferred to or specified at the boundaries must be consistent with the problem's statements (e.g., if in the problem under consideration the mother board is made of a heat-conducting material, then it is incorrect to cut the mother board with computational domain boundaries, since this will yield an incorrect heat flux from the chip through the mother board).

In this project we specify the following computational domain boundaries satisfying the above-mentioned requirements. Click **Flow Analysis > Computational Domain** to adjust the computational domain size as follows:

- X_{max} = 0.7 in (the boundary conditions in the fluid region of this boundary are transferred from the first stage's computation results, the same boundary conditions as at X_{min} = -2.95 in are automatically specified at this boundary's upper solid part lying in the electronic enclosure's aluminum wall, and the same boundary conditions as at Z_{min} = -1 in are automatically specified at the lower solid part lying in the mother board),
- X_{min} = -2.95 in (entirely lies inside the electronic enclosure side wall made of aluminum, this material does not influence the main chip's temperature since it is insulated from the chip by the heat-insulating mother board and the air flow, its boundary condition is automatically specified as the 68.09 °F temperature specified as the initial condition for all solids),
- Y_{max} = 4 in, Y_{min} = -1 in (the boundary conditions at these boundaries are specified in the same manner as at X_{max} = 0.7 in, as well as at the boundaries' side parts also lying in the aluminum wall),

- Z_{max} = 1.2 in (entirely lies inside the electronic enclosure's aluminum upper wall, therefore the same boundary condition, as at Xmin = -2.95 in, are automatically specified at this boundary),
- Z_{min} = -1.1 in (entirely lies inside the mother board specified as a heat insulator, therefore the adiabatic wall boundary condition is automatically specified at this boundary).

To adjust the computational domain size in Creo Parametric, click ***Flow Analysis > Project > Computational Domain*** .



The reduced computational domain.

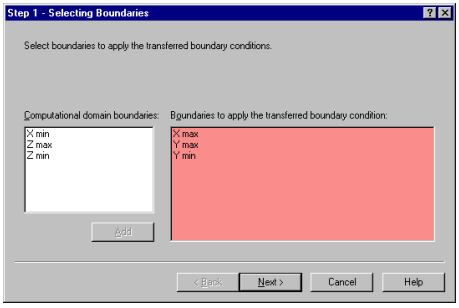
Specifying Transferred Boundary Conditions

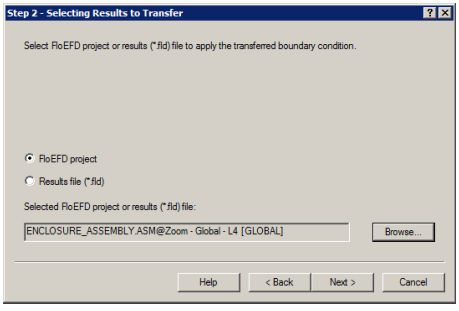
1 Click **Flow Analysis > Insert > Transferred Boundary Condition**.

2 Add the **Xmax**, **Ymax** and **Ymin Computational Domain boundaries** to the **Boundaries to apply the transferred boundary condition** list. To add a boundary, select it and click **Add**, or double-click a boundary.

3 Click **Next**.

4 At **Step 2**, click **Browse** to select the FloEFD project whose results will be used as boundary conditions for the current **Zoom – Sink 1 – L4** project.



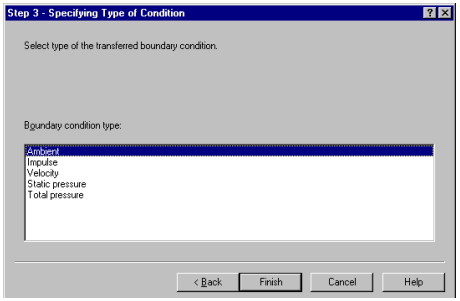

Step 2 - Selecting Results to Transfer
Select FloEFD project or results (*.fld) file to apply the transferred boundary condition.
FloEFD project
Results file (*.fld)
Selected FloEFD project or results (*.fld) file:
ENCLOSURE_ASSEMBLY.ASM@Zoom - Global - L4 [GLOBAL]
Browse...
Help
< Back
Next >
Cancel

You can select a calculated project of any currently open model, or browse for the results (.fld) file.

5 In the **Browse for Project** dialog select the **Zoom – Global – L4** project and click **OK**.

6 Click **Next**.

7 At **Step 3**, accept **Ambient** as the **Boundary condition type**.

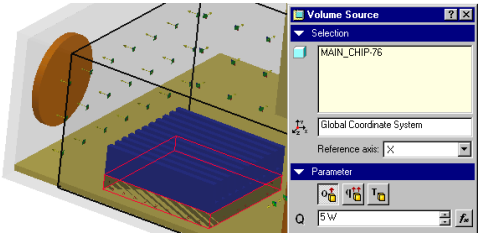


📖 *The Ambient boundary condition consists of specifying (by taking results of a previous calculation) flow parameters at the boundary's section lying in the fluid, so they will act during the calculation in nearly the same manner as ambient conditions in an external analysis. If Heat Conduction in Solids is enabled, then the solid temperature is specified at this boundary's section lying in the solid (by taking results of a previous calculation). The heat flux at this boundary, which will be obtained as part of the problem solution, can be non-zero.*

8 Click **Finish**.

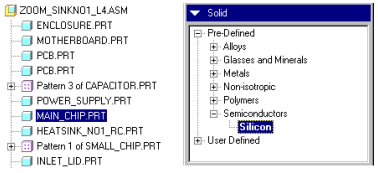
Specifying Heat Sources

Volume Source of 5W heat generation rate in the main chip.

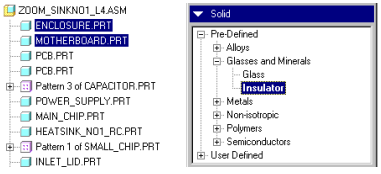


Specifying Solid Materials

a) **MAIN_CHIP** is made of silicon (Pre-Defined/Semiconductors);



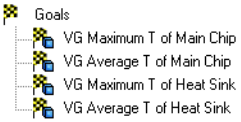
b) **MOTHERBOARD** and **ENCLOSURE** are made of insulator (Pre-Defined/Glasses & Minerals);



c) all other parts (e.g. the heat sink) are made of aluminum.

Specifying Goals

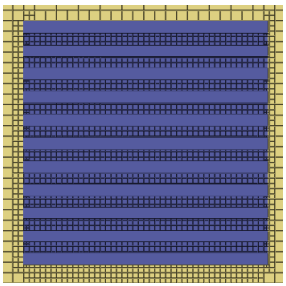
Specify the Volume Goals of maximum and average temperatures of the main chip and the heat sink.



Run the calculation.

The obtained computational results are presented in tables and pictures below. These results were obtained with the heat sink's shape N.1.

If you look at the computational mesh you can see that it has two cells for each of the heat sink's channels, and two cells for each of the sink's fins.



The mesh cut plot obtained for the heat sink No.1 at *Y=-0.3 in*.

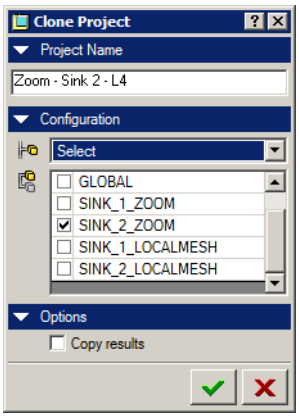
📖 *The **Minimum gap size** and **Minimum wall thickness** influence the same parameter, namely, the characteristic cell size. By default, FloEFD generates the basic mesh in order to have a minimum of two cells per the specified **Minimum gap size**. The number of cells per the **Minimum gap size** depends non-linearly on the **Level of initial mesh** and cannot be less than two. In turn, the **Minimum wall thickness** condition induces FloEFD to create the basic mesh having two cells (two cells are enough to resolve a wall) per the specified **Minimum wall thickness** (regardless of the specified initial mesh level). That's why, if the **Minimum wall thickness** is equal to or greater than the **Minimum gap size**, then the former does not influence the resulting mesh at all.*

Changing the Heat Sink Configuration

Let us now see how employing the heat sink's shape No. 2 changes the computational results. To do this, we change the heat sink configuration to the No.2 version, whereas all the EFD Zooming FloEFD project settings of 2nd stage are retained. There is no need to perform the EFD Zooming computation of 1st stage again, as we can use its results in this project too.

The easiest way to create the same FloEFD project for the new model configuration is to clone the existing project to this configuration.

Cloning Project to the Existing Configuration



1 Click **Flow Analysis > Project > Clone Project**.

2 In the **Project Name**, type `Zoom - Sink 2 - L4`.

3 In the **Configuration to Add the Project** list, select **Select**.

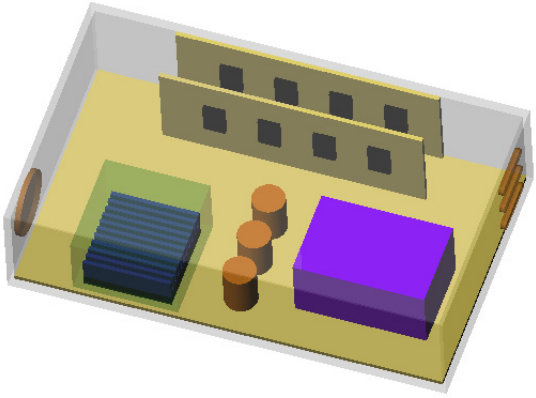
4 In the **Configurations** list, select **SINK_2_ZOOM**.

Click **OK**. After clicking OK, two warning messages appear asking you to reset the computational domain and to rebuild the computational mesh. Select **No** to ignore the resizing of computational domain, and **Yes** to rebuild the mesh.

The obtained results are presented in tables and pictures at the end of this tutorial. It is seen that due to the new shape of the heat sink the main chip's temperature is reduced by about 15 °F. That is caused by both the increased area of the heat sink's ribs and streamlining the flow in the heat sink's narrow channels between the ribs (in heat sink No.1 about half of the channel is occupied by a counterflow vortex).

The Local Initial Mesh Approach

To validate the results obtained with the EFD Zooming approach, let us now solve the same problems employing the Local Initial Mesh option. To employ this option, we add a parallelepiped surrounding the main chip to the model assembly and then disable it in the **Component Control** dialog box. This volume represents a fluid region in which we can specify computational mesh settings differing from those in the other computational domain, using the **Local Initial Mesh** option.



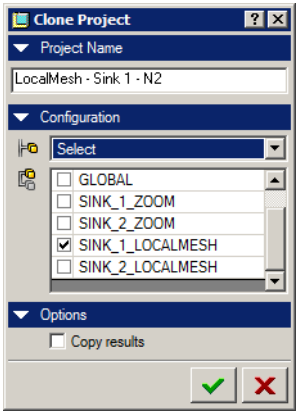
The electronic enclosure configuration with the additional part for applying the Local Initial Mesh option.

FloEFD Project for the Local Initial Mesh Approach (Sink No1)

To create the project in this case, we clone the **Zoom - Sink 1 - L4** project to the existing **SINK_1_LOCALMESH** configuration, but in contrast to the previous cloning, we reset the computational domain to the default size so the computational domain encloses the entire model.

Activate **Zoom - Sink 1 - L4** project.

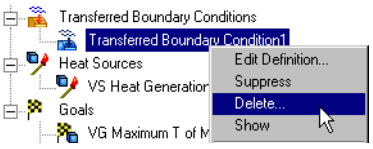
Open the **Clone Project** dialog, in the **Project Name**, type `LocalMesh - Sink 1 - N2`. In the **Configuration to Add the Project** list, select **Select**. In the **Configurations** list, select **SINK_1_LOCALMESH** as the configuration to which FloEFD will attach the cloned project.



After clicking **OK**, confirm with **Yes** both messages appearing.

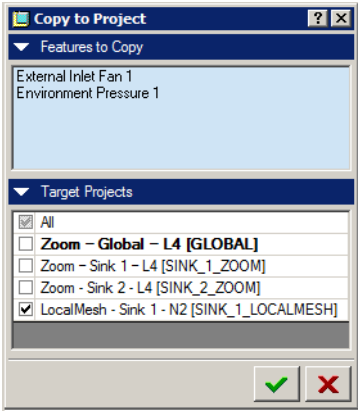
Specifying Boundary Conditions

First remove the inherited transferred boundary condition. Right-click the **Transferred Boundary Condition1** item in the tree and select **Delete**.



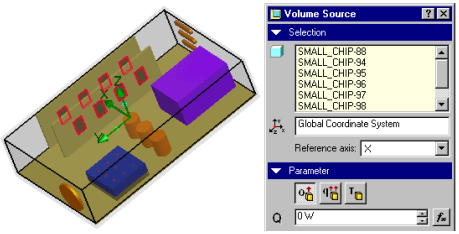
Next, copy the boundary conditions from the **Zoom - Global - L4** project using the **Copy to Project** tool.

1. Activate **Zoom- Global - L4** project.
2. Click **Flow Analysis > Tools > Copy to Project**. The **Copy to Project** dialog box appears.
3. Hold down the **Ctrl** key and in the FloEFD Analysis tree select **Environment Pressure 1** and **External Inlet Fan 1** items. These features appear in the **Features to copy** list.
4. Select **LocalMesh - Sink 1 - N2** as the **Target Project**.
5. Click **OK**.
6. Activate **LocalMesh - Sink 1 - N2** project.


Copy to Project
Features to Copy
External Inlet Fan 1
Environment Pressure 1
Target Projects
All
Zoom – Global – L4 [GLOBAL]
Zoom – Sink 1 – L4 [SINK_1_ZOOM]
Zoom - Sink 2 - L4 [SINK_2_ZOOM]
LocalMesh - Sink 1 - N2 [SINK_1_LOCALMESH]

Specifying Heat Sources

To the already existing volume source of the 5W specified in the main chip, add a 5W volume heat generation source applied on the small chips.



Specifying Solid Materials

The following material definitions were inherited from the previous project so you do not need to create them again, but you need to edit the **Silicon Solid Material 1** to include small chips and to edit **Insulator Solid Material 1** to include inlet, outlet and screwhole lids:

a) the **MAIN_CHIP** and small chips are made of silicon;

b) the **MOTHERBOARD**, the **ENCLOSURE**, the **INLET_LID** and the **OUTLET_LID**s are made of insulator;

c) Both small **PCB**s are made of user defined **Tutorial PCB** material, which is added to the Engineering Database in the **A2 - Conjugate Heat Transfer** tutorial example.

d) all other parts are made of the default aluminum.

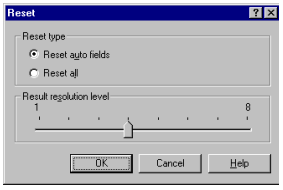
Specifying Goals

Keep the cloned volume goals of maximum and average temperatures of the main chip and the heat sink.

Changing the Level of Initial Mesh

Click **Flow Analysis > Initial Mesh** to adjust the automatic initial mesh settings.

*To adjust the automatic initial mesh settingsin Creo Parametric, click **Flow Analysis > Mesh > Initial Mesh**.*

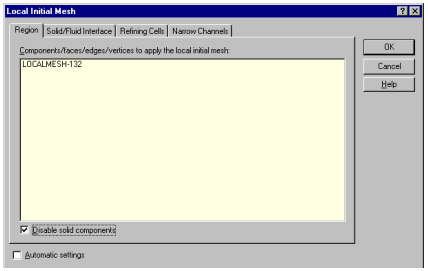


Set the **Level of initial mesh** to 3. Since heat conduction in solids is enabled, setting the **Level of initial mesh** to 4 together with the local mesh settings will produce large number of cells resulting in longer CPU time. To decrease the calculation time for this tutorial example we decrease the **Level of initial mesh** to 3. Note that the **Result resolution level** is still equal to 4 as it was specified in the Wizard. To see the value of the result resolution level, click **Flow Analysis > Calculation Control Options**, and then click **Reset.** To close the **Reset** dialog box, click **Cancel**.

Click **Flow Analysis > Project > Rebuild**.

Specifying Local Initial Mesh Settings

To apply the local mesh setting to a region we need a component representing this region to be disabled in the **Component Control** dialog box.



1 Click **Flow Analysis > Insert > Local Initial Mesh**.

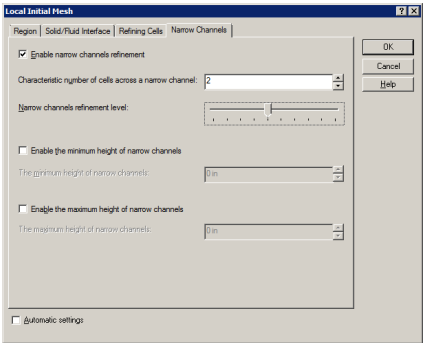
✎ *In Creo Parametric click **Flow Analysis > Mesh > Local Initial Mesh**.*

2 In the Model Tree, select the **LocalMesh** component.

3 Click the **Disable solid components** check box.

4 Clear the **Automatic settings** check box.

5 Go to the **Narrow Channels** tab and set the **Characteristic number of cells across a narrow channel** = *2* and **Narrow channels refinement level** = *4*.



🕮 *The **Narrow Channels** term is conventional and used for the definition of the model's flow passages in the normal-to-solid/fluid-interface direction. The procedure of refinement is applied to each flow passage within the computational domain unless you specify for FloEFD to ignore the passages of a specified height with the **Enable the minimum height of narrow channels** and **Enable the maximum height of narrow channels** options. The **Characteristic number of cells across a narrow channel** (let us denote it as **Nc**) and **Narrow channels refinement level** (let us denote it as **L**) both influence the mesh in narrow channels in the following way: the basic mesh in narrow channels will be split to have the specified **Nc** number per a channel, if the resulting cells satisfy the specified **L**. In other words, whatever the specified **Nc**, a narrow channel's cells cannot be smaller in 8^L (2^L in each direction of the Global Coordinate System) times than the basic mesh cell. This is necessary to avoid the undesirable mesh splitting in superfine channels that may cause increasing the number of cells to an excessive value.*

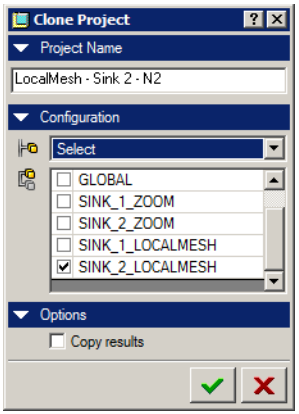
6 Click **OK**.

In our case, to ensure the 2 cells across a channel criterion, we increased the **Narrow channels refinement level** to 4.
We perform these settings for both of the heat sinks under consideration.

FloEFD Project for the Local Initial Mesh Approach (Sink No2)

Clone the active **LocalMesh – Sink 1 – N2** project to the existing **SINK_2_LOCALMESH** configuration (instance). Name the new project **LocalMesh - Sink 2 - N2**. While cloning confirm the message to rebuild the mesh.

Using the **Batch Run**, calculate both projects.

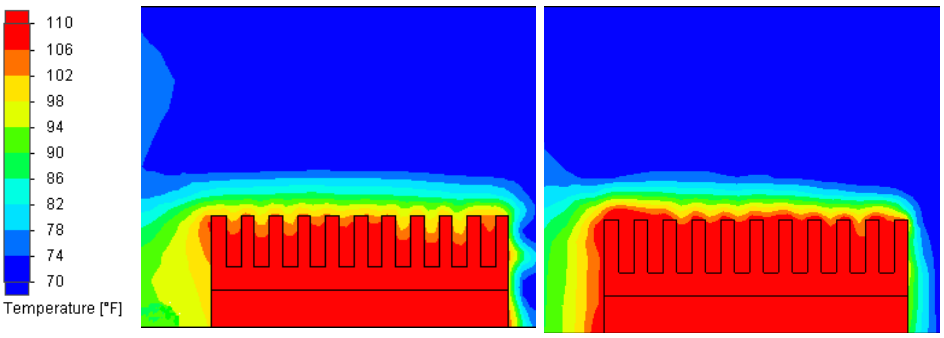


Results

The computational results obtained for both of the heat sinks are presented below in comparison with the results obtained with the EFD Zooming approach. It is seen that computations with the local mesh settings yield practically the same results as the EFD Zooming approach.

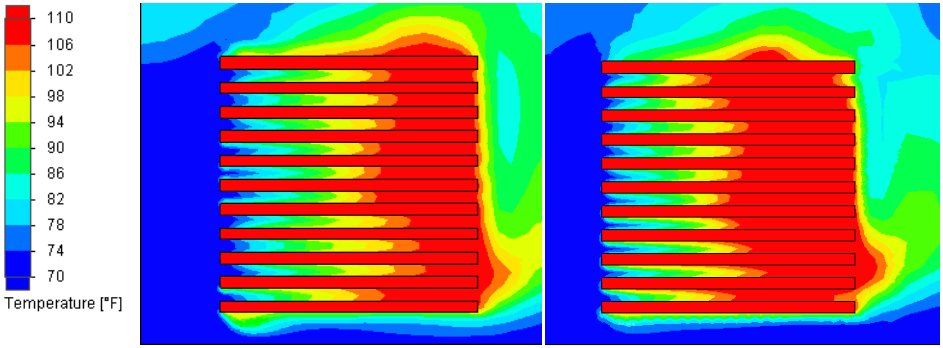
Parameter		Heat sink No.1		Heat sink No.2	
		Zoom	LocalMesh	Zoom	LocalMesh
Main chip	t_{max}, °F	110,6	114,5	96,8	99,4
	t_{aver}, °F	110,3	114,2	96,6	99,2
Heat sink	t_{max}, °F	110,6	114,5	96,7	99,4
	t_{aver}, °F	110,2	114	96,4	99

EFD Zooming **Local Mesh**

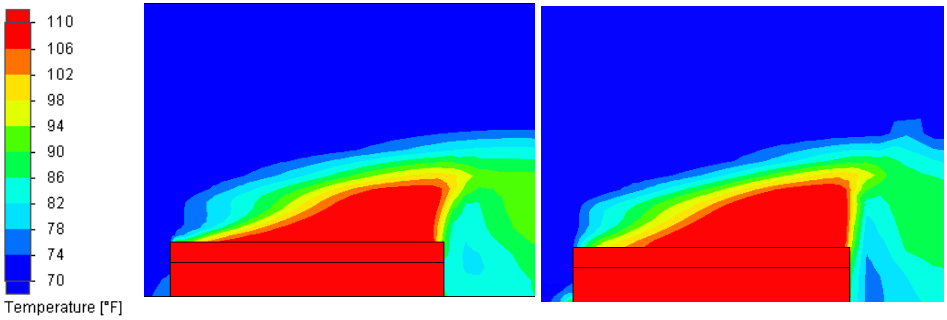


EFD Zooming **Local Mesh**

The temperature cut plots obtained for heat sink No.1 at *Y=2.19* in Top plane with the EFD Zooming (left) and Local Mesh (right) approaches.

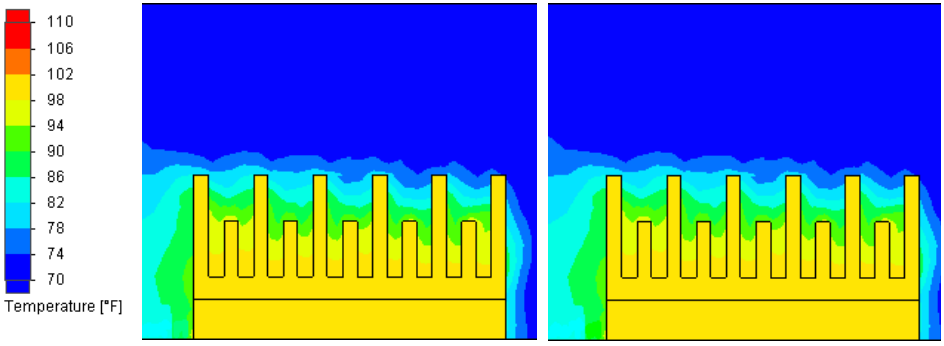


The temperature cut plots obtained for heat sink No.1 at *X= -1.53* in Right plane with the EFD Zooming (left) and Local Mesh (right) approaches.

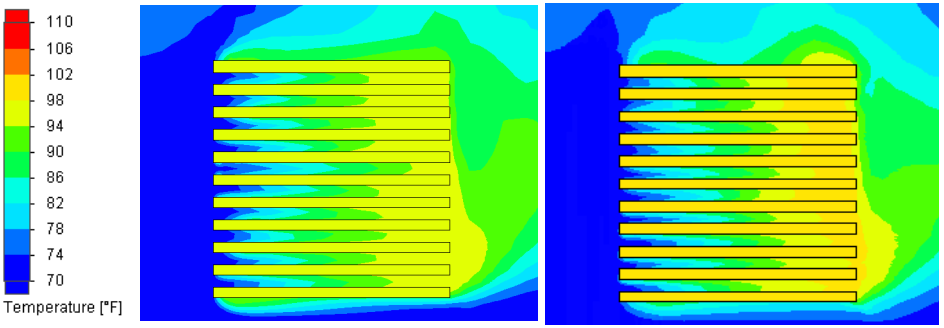


EFD Zooming **Local Mesh**

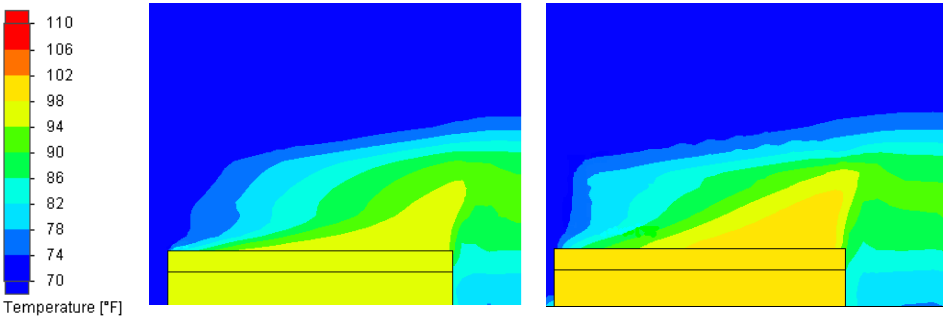
The temperature cut plots obtained for heat sink No.2 at *Y=2.19* in Top plane with the EFD Zooming (left) and Local Mesh (right) approaches.



EFD Zooming	Local Mesh



The temperature cut plots obtained for heat sink No.2 at *Z= -0.32* in Front plane with the EFD Zooming (left) and Local Mesh (right) approaches.



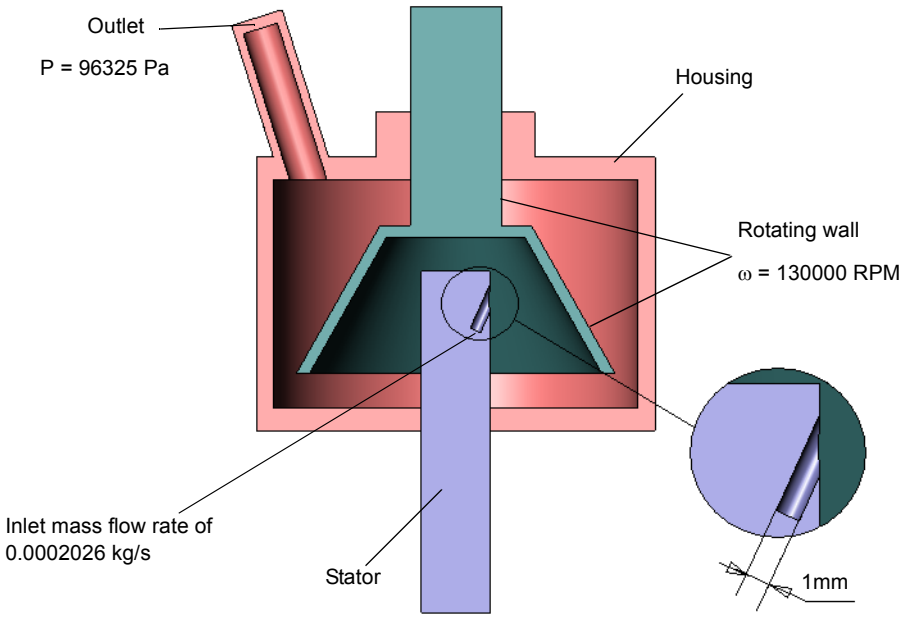
The temperature cut plots obtained for heat sink No.2 at *X= -1.53* in Right plane with the EFD Zooming (left) and Local Mesh (right) approaches.

C2

Textile Machine

Problem Statement

The simplified textile machine used in this tutorial is described as a closed hollow cylinder having a cylindrical stator with a narrow inlet tube (see the figure below). A thin-walled cone rotates at a very high speed. The air flows over the rotating cone before leaving through the outlet pipe. Due to the shear stress, the rotating cone swirls the air. The swirling air motion orients the fibers, for the correct formation of yarn.



In this example a hollow cylinder with the following dimensions were used: 32 mm inner diameter and 20 mm inner height. Air is injected into an inlet tube of 1 mm diameter at a mass flow rate of 0.0002026 kg/s. The cone thickness is 1 mm and the cone's edge is spaced at 3 mm from the bottom of the main cylinder. The cone rotates at a speed of 130000 RPM. The static pressure of 96325 Pa is specified at the cylinder's outlet tube exit.

FloEFD analyzes the air flow without any fiber particles. The influence of the fiber particles on the air flow was assumed to be negligible. Small polystyrene particles were injected into the air stream using the results processing Flow Trajectory feature to study the air flows influence on the fibers. A 40 m/s tangential velocity of air is specified as an initial condition to speed up convergence and reduce the total CPU time needed to solve the problem.

Opening the Model

Copy the **C2 - Textile Machine** folder into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files. Open the **textile_machine.asm** assembly.

🕮 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **textile_machine.asm** assembly located in the **C2 - Textile Machine\Ready To Run** folder and run the desired projects.*

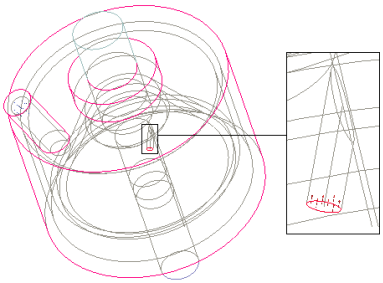
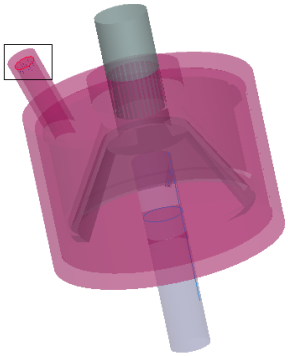
Creating a FloEFD Project

Using the **Wizard** create a new project as follows:

Project name	*130000rpm*
Configuration	*Use current*
Unit system	**SI**; *select* **mm** *(***Millimeter***) for* **Length, RPM** *(***Rotations Per Minute***) for* **Angular Velocity** *and* **kg/h** *(***Kilogram/hour***) for* **Mass Flow Rate** *under* **Loads&Motion**
Analysis type	**Internal**; **Exclude cavities without flow conditions**
Physical features	*No physical features are selected*
Fluid	**Air**
Wall Conditions	*Adiabatic wall, Zero roughness*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	**Result resolution level** *set to* 4; **Minimum gap size** = 1 mm, *other options are default*

Specifying Boundary Conditions

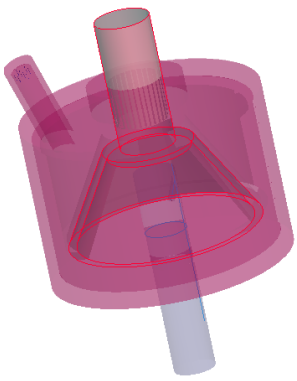
Specify the inlet and outlet boundary conditions as follows:

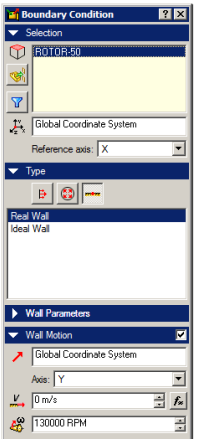
Inlet Boundary Condition	***Inlet Mass Flow = 0.73 kg/h:*** *Inlet mass flow rate of 0.73 kg/h normal to the inlet face of the **STATOR**;*	
Outlet Boundary Condition	***Outlet Static Pressure = 96325 Pa:*** *Static pressure of 96325 Pa at the outlet face of the **HOUSING** (the other parameters are default).*	

Specifying Rotating Walls

The influence of parts and components rotation on the flow can be simulated in FloEFD in two ways. With the Rotating Region feature you can assign a rotating reference frame to a selected fluid region. This allows to simulate the rotation of components of complex geometry, such as fans, pump wheels, impellers, etc. In this tutorial we consider rotation of a component with a relatively simple geometry. All surfaces of the textile machine rotor are surfaces of revolution such as cones or cylinders. For this kind of rotating geometry the Moving Wall boundary condition is better suited and usually provides more accurate results.

1. In the FloEFD Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.
2. Select **Wall**, then **Real Wall**.
3. In the Model Tree select the **ROTOR** component.
4. Select **Wall Motion**.
5. Select **Y** as the rotation **Axis**.
6. Specify the **Angular Velocity** of 130000 RPM.
7. Click **OK** and rename the new **Real Wall 1** item to `Rotating Wall = 130 000 rpm`.





Specifying Initial Conditions

In order to speed up the convergence, a 40 m/s tangential velocity of air is specified as an initial condition within the housing. The **INITIAL_VELOCITY_1** and **INITIAL_VELOCITY_2** auxiliary components are used to define a fluid domain.

1 Click **Flow Analysis > Insert > Initial Condition**.

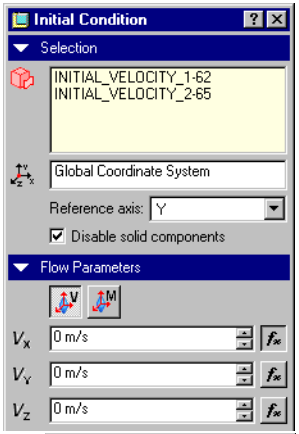
2 In the Model Tree select the **INITIAL_VELOCITY_1** and **INITIAL_VELOCITY_2** components.

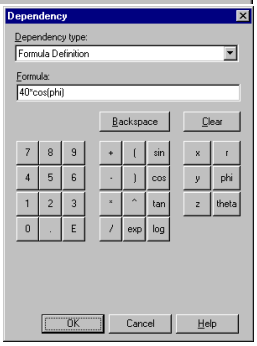
3 Select the **Disable solid components** option. FloEFD will treat these components as a fluid region.

4 Select **Y** in the **Reference axis** list.

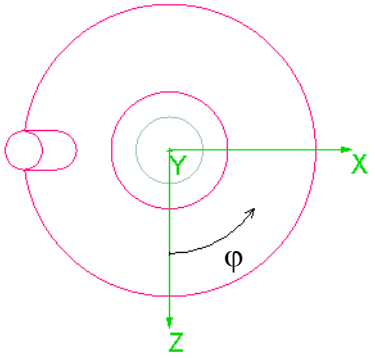
5 Under **Flow Parameters**, select the **Dependency** for the **Velocity in X direction** by clicking the f_x button. The **Dependency** dialog box appears.

6 In the **Dependency type** list, select **Formula Definition**.





7 In the **Formula** box, type the formula defining the velocity in X direction: `40*cos(phi)`.
Here ***phi*** is the polar angle φ defined as shown on the picture below.

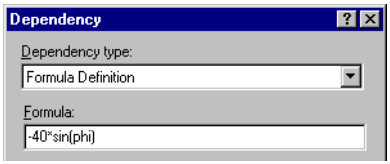


8 Click **OK**. You will return to the **Initial Condition** dialog box.

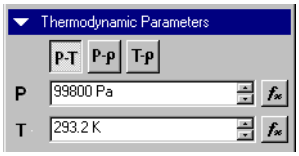
9 Click in the **Velocity in Z direction** item.

10 Click **Dependency** and specify formula for the Z component of velocity: `-40*sin(phi)`.

11 Click **OK**.



12 Under **Thermodynamic Parameters,** change the **Pressure** P to `99800` Pa.



13 Click **OK** .

14 Click-pause-click the new **Initial Condition1** item and rename it to `vel = 40`.

Specifying Goals

Since the rotating cone swirls the air, it make sense to specify the air velocity as a goal to ensure the calculation stops when the velocity is converged. In addition, let us specify the static pressure surface goal at the inlet and the mass flow rate surface goal at the outlet as additional criteria for converging the calculation.

Specify the following project goals:

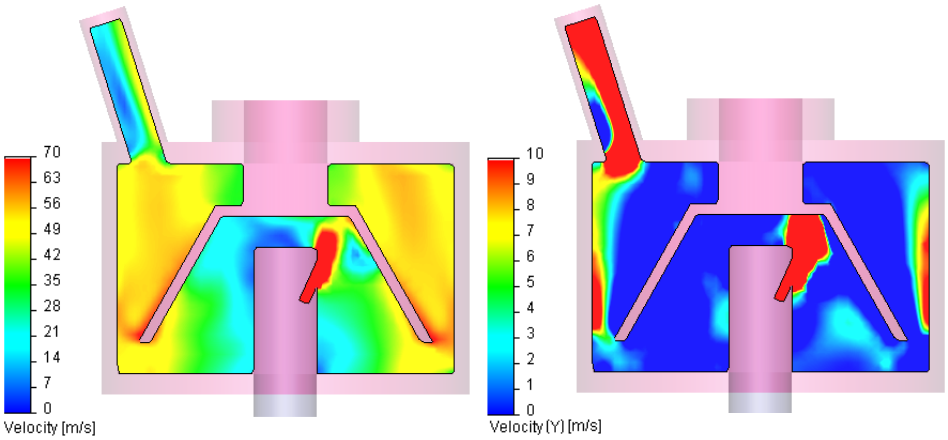
GOAL TYPE	GOAL VALUE	FACE/COMPONENT
Global Goal	Average Velocity	
Surface Goal	Mass Flow Rate	Outlet face (click the outlet static pressure boundary condition item to select the outlet face)
Surface Goal	Average Static Pressure	Inlet face (click the inlet mass flow rate boundary condition item to select the inlet face)
Volume Goal	Average Velocity	**INITIAL_VELOCITY_1** (select the component in the Model Tree)
Volume Goal	Average Velocity	**INITIAL_VELOCITY_2** (select the component in the Model Tree)

Calculate the project.

Results (Smooth Walls)

The calculated flow velocity field and velocity Y-component field at Z = 0 (XY section) are shown in the pictures below. It can be seen that the maximum flow velocity occurs near the inlet tube and near the rotating cone's inner surface at the cone's edge.

Velocity in the XY section at Z = 0.



Flow velocity

Y-component of flow velocity

It is interesting that the vertical (i.e. along the Y axis) velocity in the region close to the rotating cone's internal and external surfaces is directed to the cylinder bottom. Also, this velocity component is nearly zero in the gap between the rotating cone and the bottom of the cylinder, and positive (i.e. directed to the top) in the vicinity of the cylinder's side walls. As a result, small particles carried by the air into the region between the lower edge of the rotating cone and the bottom of the cylinder cannot leave this region due to the small vertical velocity there. On the other hand, larger particles entering this region may bounce from the cylinder's bottom wall (in this example the ideal, i.e. full reflection is considered) and fly back to the region of high vertical velocity. Then they are carried by the air along the cylinder's side walls to the cylinder's top wall where they remain in this region's vortex.

Displaying Flow and Particles Trajectories

To display flow trajectories as flow streamlines, we need to specify the starting points through which the trajectory passes and the streamline direction relative to these points.

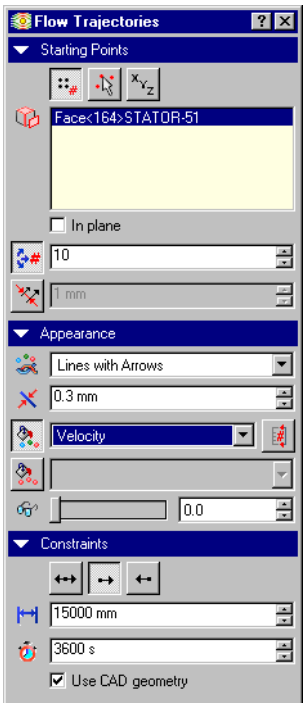
1 In the FloEFD Analysis tree, right-click the **Flow Trajectories** icon and select **Insert**.

2 In the FloEFD Analysis tree click the inlet boundary condition icon (**Inlet Mass Flow = 0.73 kg/h**) to select the corresponding face.

3 Set the **Number of Points** to `10`.

4 Under **Appearance**, set the **Draw Trajectories As** to **Lines with Arrows** and change **Color by Parameter** to **Velocity**.

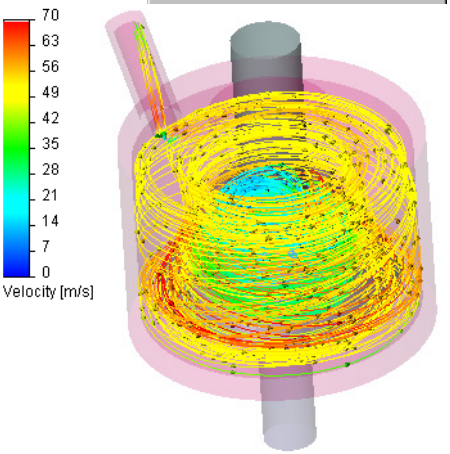
5 Under **Constraints.** select the **Forward** direction and increase the **Maximum Length** of trajectories to `15` m.



*The **Maximum length** option limits the length of the trajectory to the specified value. We increase this value to show better the flow vorticity.*

6 Click **OK** to display flow streamlines.

7 To specify the parameter display range click the maximum value in the velocity palette bar and type `70 m/s` in an edit box.

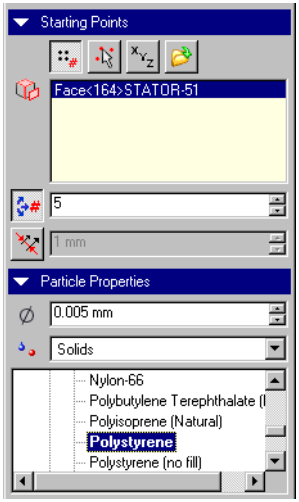


To display particles trajectories, we need to specify initial particle properties (temperature, velocity and diameter), particle's material and the wall condition (absorption or reflection).

1 In the Analysis tree, right-click the **Particle Studies** icon and select **Wizard**.

2 Keep the default name for the Particle Study and click **Next**.

3 Click the inlet boundary condition icon (**Inlet Mass Flow = 0.73 kg/h**) in the tree to select the inlet face from which the particles are injected.



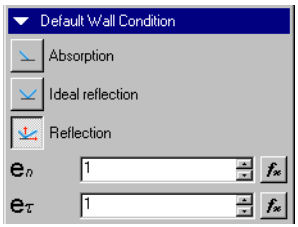
4 Under the **Starting Points**, set the **Number of Points** to 5.

5 Under the **Particle Properties**, set the **Diameter** equal to `0.005 mm` and change the **Material** to the **Solids** and in the list select the **Polystyrene** (**Materials** / **Solids** / **Pre-Defined** / **Polymers**).

We leave unchanged the default zero values of relative velocity and temperature, which means that the velocity and temperature of particles are equal to those of the incoming flow. We also leave the default value of mass flow rate, since it is used only to estimate mass rates of erosion or accumulation, which we are not going to take into account.

6 Click **Next** twice.

7 Change the **Default Wall Condition** to **Reflection**.



8 Click **Next**.

9 Under **Default Appearance**, set **Draw Trajectories as Lines with Arrows**.

10 Under **Constraints,** increase the **Maximum Length** of trajectories to `15 m`.

11 Click **OK**. A new **Particle Study 1** item with one sub-item (**Injection 1**) appear in the Analysis tree.

12 Right-click the created **Injection 1** item and select **Clone**. The **Injection 2** item will be created. For this item, increase the particle size by editing the **Diameter** to `0.015 mm`.

13 Right-click the **Particle Study 1** item and select **Run**.

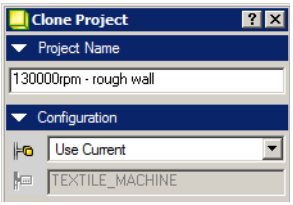
14 Select **Injection 1** and click **Show** to view the particles' trajectories.

15 When finished examining the trajectories from the first injection, hide the **Injection 1** trajectories and show the **Injection 2** trajectories.

Modeling Rough Rotating Wall

In the previous calculation zero roughness was used for the walls of the rotating cone's internal and external surfaces. To investigate an influence of the rotating cone wall's roughness, let us perform the calculation with the rotating cone's internal and external surfaces' at 500 μm roughness under the same boundary conditions.

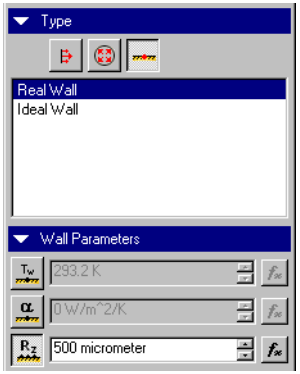
Create a new project by cloning the current project to the current configuration, and name it `130000rpm - rough wall`.



Adjusting Wall Roughness

1. Right-click the **Rotating Wall = 130 000 rpm** item and select **Edit Definition**.
2. Under **Wall parameters**, select **Adjust Wall Roughness** .
3. Specify the wall roughness of `500 micrometers`.
4. Click **OK**.

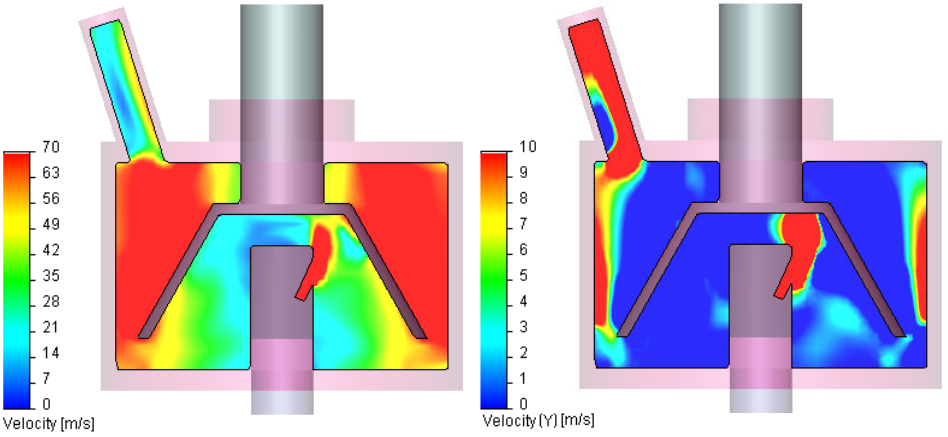
Run the calculation.



Results (Rough Walls)

The calculated fields of flow velocity and Y-component of velocity in different section are shown below and reveal practically no change in the vertical velocity of the flow. As a result, the flying particles' trajectories are nearly identical to those in the case of smooth walls. It is seen that increase in the roughness from 0 to 500 μm increases the vortex flow's tangential velocity.

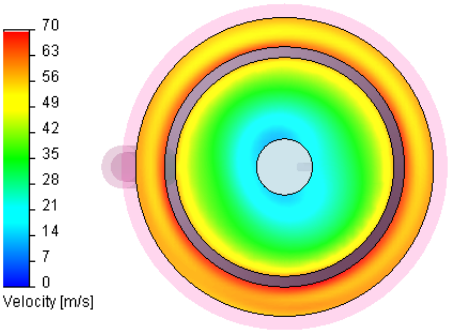
Velocity in the XY section at Z = 0 (roughness = 500 μm)



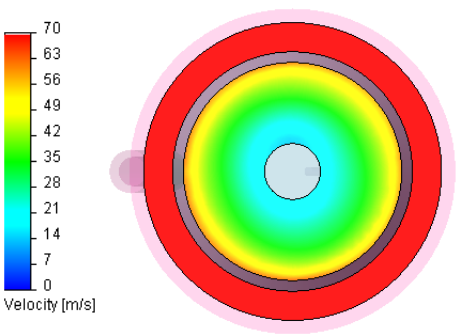
Flow velocity

Flow velocity's Y-component

Velocity in the ZX section at Y = 2 mm

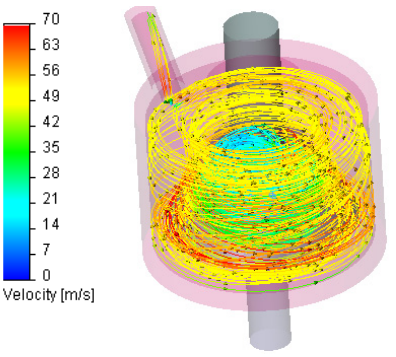


roughness = 0 μm

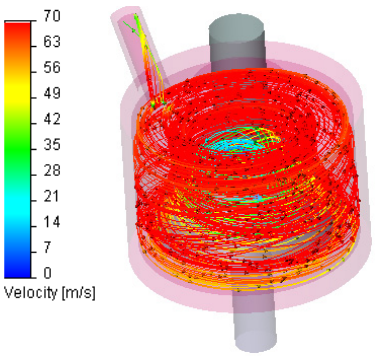


roughness = 500 μm

Flow trajectories

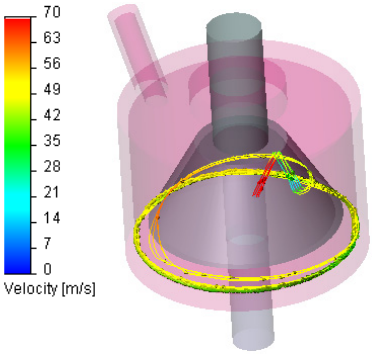


Smooth wall

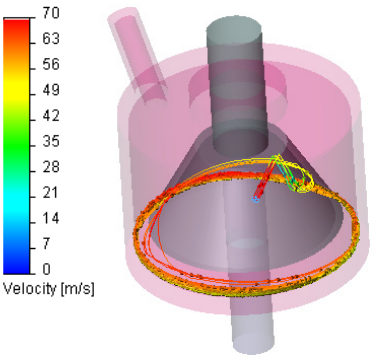


Rough wall

Trajectories of 5 μm particles

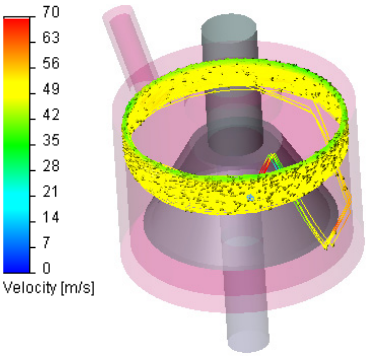


Smooth wall

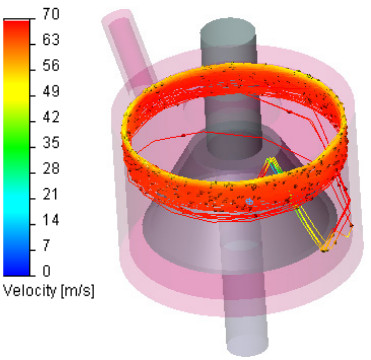


Rough wall

Trajectories of 15 μm particles



Smooth wall



Rough wall

C3

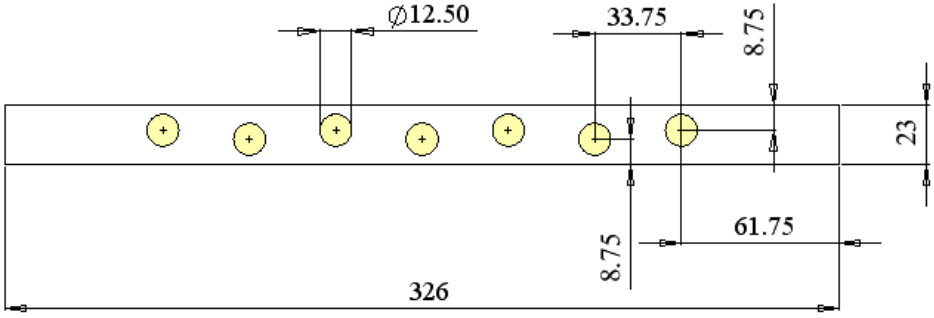
Non-Newtonian Flow in a Channel with Cylinders

Problem Statement

Let us consider a non-Newtonian liquid's 3D flow through a rectangular-cross-section channel encumbered with seven circular cylinders arranged asymmetrically with respect to the channel's midplane shown in **Ref.1**. Following **Ref.1**, let us consider the 3% aqueous solution of xanthan gum as a non-Newtonian liquid. Its viscosity approximately obeys the power law $\eta = K \cdot \left(\dot{\gamma}\right)^{n-1}$ with a consistency coefficient of K = 20 Pa×s^n and a power-law index of $n = 0.2$, whereas its other physical properties (density, etc.) are the same as in water (since the solution is aqueous).

The problem's goal is to determine the total pressure loss in the channel. Also, to highlight the influence of the 3% xanthan gum addition to water on the channel's total pressure loss, we will calculate the flow of water using the same volume flow rate within the channel.

The FloEFD calculations are performed with the uniform liquid velocity profile at the channel inlet, the liquid's volume flow rate is 50 cm^3/s. The static pressure of 1 atm is specified at the channel outlet. The calculation's goal is the channel's resistance to the flow, i.e., the total pressure drop ΔP_o between the channel inlet and outlet.



Opening the Model

Copy the **C3 - Non-Newtonian Flow** folder into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files. Open the **array_of_cylinders.asm** assembly.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **array_of_cylinders.asm** assembly located in the **C3 - Non-Newtonian Flow\Ready To Run** folder and run the desired projects.*

Defining Non-Newtonian Liquid

1. Click **Flow Analysis > Tools > Engineering Database**.
2. In the **Database tree**, select **Materials** / **Non-Newtonian Liquids / User Defined**.
3. Click **New Item** in the toolbar. The blank **Item Properties** tab appears. Double-click the empty cell to set the corresponding property value.
4. Specify the material properties as shown in the table below:

Name	XGum
Density	1000 kg/m^3
Specific heat	4000 J/(kg*K)
Thermal conductivity	0.6 W/(m*K)
Viscosity	Power law model
Consistency coefficient	20 Pa*s^n
Power law index	0.2

Save and exit the database.

Project Definition

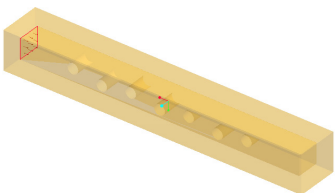
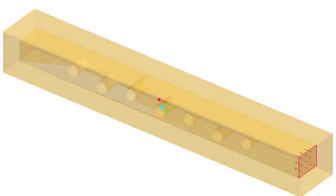
Using the **Wizard** create a new project as follows:

Project name	*XGS*
Configuration	*Use current*
Unit system	**CGS** *modified:* **Pa** *(***Pascal***) for the* **Pressure & Stress**

Analysis type	**Internal**; **Exclude cavities without flow conditions**
Physical features	*No physical features are selected (default)*
Fluid	**XGum (non-Newtonian liquids)**; *Flow type: Laminar only (default)*
Wall Conditions	*Adiabatic wall, default smooth walls, default slip condition*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Default result resolution level 3;* **Minimum gap size**=0.25 cm, *other options are default*

Specifying Boundary Conditions

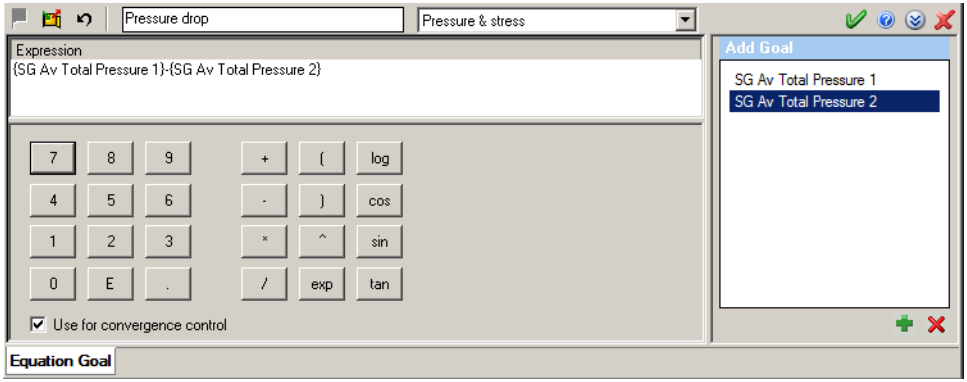
Specify boundary conditions as follows:

Inlet Boundary Condition	***Inlet Volume Flow 1:*** *50 cm^3/s Volume flow rate; default temperature (20.05 °C) at the face shown in the figure;*	
Outlet Boundary Condition	***Static Pressure 1***: *Default value (101325 Pa) for the Static pressure at the face shown in the figure;*	

Specifying Goals

Specify surface goals for the **Average Total Pressure** at the inlet and outlet.

Specify an equation goal for the total pressure drop between the channel's inlet and outlet.



Run the calculation. When the calculation is finished, create the goal plot to obtain the pressure drop between the channel's inlet and outlet.

ARRAY_OF_CYLINDERS.ASM [XGS]

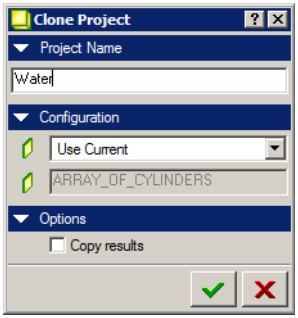
Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]
SG Av Total Pressure 1	[Pa]	105652,2964	105652,217	105649,8632	105655,001	100
SG Av Total Pressure 2	[Pa]	101329,319	101329,3176	101329,3116	101329,3199	100
Pressure Drop	[Pa]	4322,977406	4322,899403	4320,548595	4325,681399	100

It is seen that the channel's total pressure loss is about 4 kPa.

Comparison with Water

Let us now consider the flow of water in the same channel under the same conditions (at the same volume flow rate).

Create a new project by cloning the current project to the current configuration, and name it `Water`.



Changing Project Settings

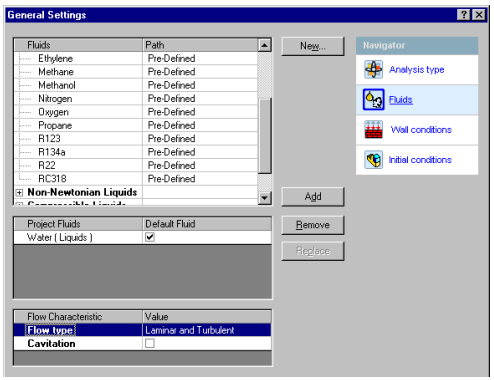
1 Click **Flow Analysis > General Settings**.

*In Creo Parametric click **Flow Analysis > Project > General Settings**.*

2 On the **Navigator** click **Fluids**.

3 In the **Project Fluids** table, select **XGum** and click **Remove**.

4 Select **Water** in **Liquids** and click **Add**.



5 Under **Flow Characteristics**, change **Flow type** to **Laminar and Turbulent**.

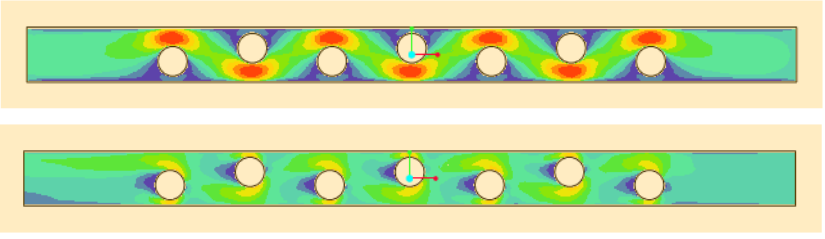
6 Click **OK**.

Run the calculation. After the calculation is finished, create the goal plot.

ARRAY OF CYLINDERS.ASM [WATER]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]
SG Av Total Pressure 1	[Pa]	101404,1827	101403,8876	101403,3704	101404,3257	100
SG Av Total Pressure 2	[Pa]	101329,9385	101329,9622	101329,9138	101330,0171	100
Pressure Drop	[Pa]	74,24418023	73,92542066	73,36800469	74,37034857	100

As shown in the results table above, the channel's total pressure loss is about 60 Pa, i.e. 60...70 times lower than with the 3% aqueous solution of xanthan gum, this is due to the water's much smaller viscosity under the problem's flow shear rates.



The XGS (above) and water velocity distribution in the range from 0 to 30 cm/s.

1 Georgiou G., Momani S., Crochet M.J., and Walters K. *Newtonian and Non-Newtonian Flow in a Channel Obstructed by an Antisymmetric Array of Cylinders*. Journal of Non-Newtonian Fluid Mechanics, v.40 (1991), p.p. 231-260.

C4

Radiative Heat Transfer

Problem Statement

Let us consider a ball with diameter of 0.075 m, which is continuously heated by a 2 kW heat source. The ball radiates heat to a concentrically arranged hemispherical reflector with the inner diameter of 0.256 m, and through a glass cover of the same inner diameter to a circular screen with the diameter of 3 m arranged coaxially with the reflector at the 1 m distance from the ball. All parts except the glass cover are made of stainless steel. The ball surface and the screen surface facing the ball are blackbody. The other side of the screen side is non-radiating. The goal of the simulation is to see how the presence of reflector and its emissivity influence the ball and screen temperatures. To do that, the following three cases are considered:

- Case 1: the reflector inner surface is whitebody;
- Case 2: all reflector surfaces are blackbody;
- Case 3: the reflector is removed.

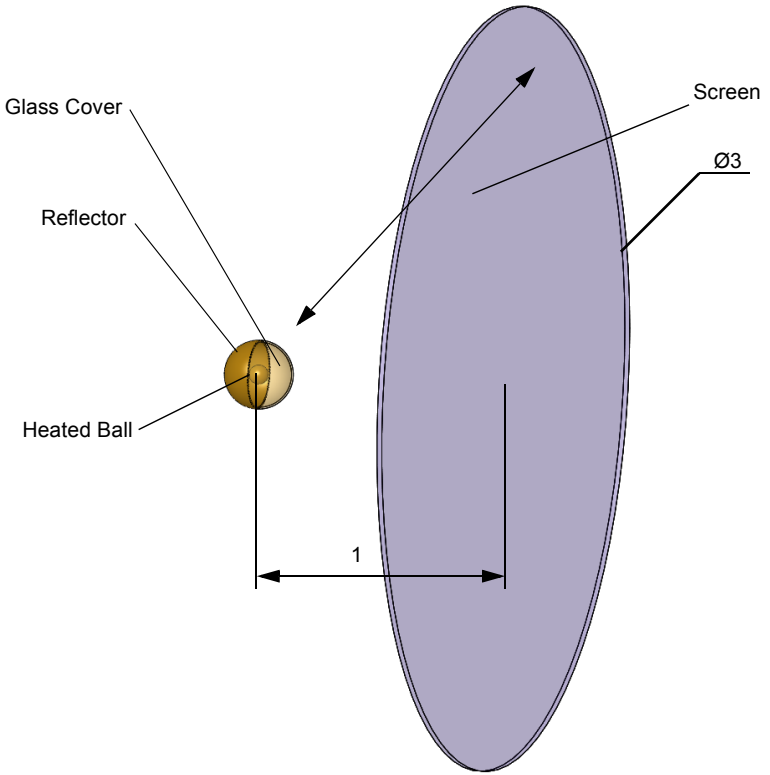
The steady-state problem is solved with the **Heat conduction in solids** option selected, so that conduction within all parts is calculated. Considering the convective heat transfer negligibly low (as in highly rarefied air), we also select the **Heat conduction in solids only** option. With this option, we do not need to specify a fluid for the project, and it is calculated without considering any fluid flow at all, thus saving the CPU time and limiting the heat transfer between parts to radiation only. The initial temperature of the parts is assumed to be 293.2 K.

Let us consider the solutions obtained with FloEFD for each of the cases under consideration.

Opening the Model

Copy the **C4 - Radiative Heat Transfer** folder into your working directory and ensure that the files are not read-only since FloEFD will save input data to these files. Open the **heated_ball_assembly.asm** assembly.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **heated_ball_assembly.asm** assembly located in the **C4 - Radiative Heat Transfer\Ready To Run** folder and run the desired projects.*



The heated ball with the reflector and screen.

Case 1: The reflector inner surface is a whitebody

Creating a FloEFD Project

Using the **Wizard** create a project as follows:

Project name	*Case 1*
Configuration	*Use current*
Unit system	**SI**
Analysis type	**External**
Physical features	**Heat conduction in solids**, **Heat conduction in solids only**, **Radiation**, **Radiation model: Discrete Transfer** **Environment temperature** = `293.2 K`;
Default Solid	**Alloys / Steel Stainless 321**
Wall conditions	**Default wall radiative surface: Non-radiating surface**;
Initial and Ambient Conditions	**Default initial solid temperature** *of* `293.2 K`
Result and Geometry Resolution	*Default result resolution level of 3;* **Minimum wall thickness** = `0.007 m`; *other options are default.*

Adjusting the Computational Domain size

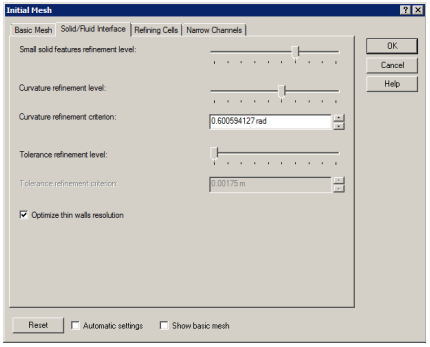
Specify the computational domain size as follows:

X max = 1.4 m	Y max = 1.6 m	Z max = 1.6 m
X min = -0.2 m	Y min = -1.6 m	Z min = -1.6 m

Adjusting Initial Mesh Settings

Click **Flow Analysis > Initial Mesh**. Clear the **Automatic settings** check box to switch off the automatic initial mesh settings, switch to the **Solid/Fluid Interface** tab and change the **Curvature refinement level** to 5. Click **OK**.

*To adjust Initial Mesh settings in Creo Parametric, click **Flow Analysis > Mesh > Initial Mesh**.*

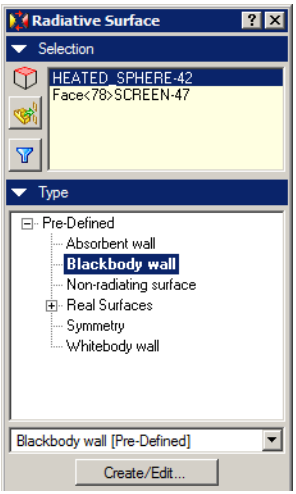


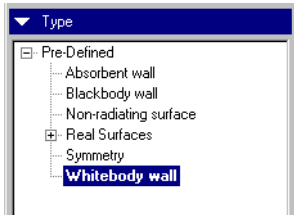
Specifying Radiative Surfaces

Follow the steps below to specify the radiative surfaces:

1. Click **Flow Analysis > Insert > Radiative Surface**.
2. Under **Type**, expand the list of **Pre-Defined** radiative surfaces and select **Blackbody wall**.
3. In the Model Tree select the **HEATED_SPHERE** component. Next, holding the **CTRL** key, select the surface of **SCREEN** facing the **HEATED_SPHERE**.
4. Click **OK**. Rename the new **Radiative Surface 1** item to `Blackbody Walls`.

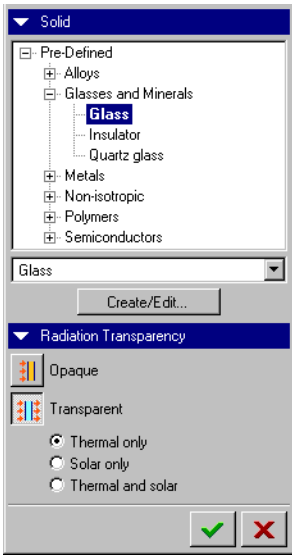
   Click anywhere in the graphic area to clear the selection.
5. Click **Flow Analysis > Insert > Radiative Surface**.
6. Select the inner surface of **REFLECTOR**.
7. Under **Type**, expand the list of **Pre-Defined** radiative surfaces and select **Whitebody wall**.
8. Click **OK**. Change the name of the new radiative surface to `Whitebody Wall`.





Specifying Bodies and Materials Transparent to the Heat Radiation

Assign the **Glass** material to the glass cover and specify it as transparent to radiation.



1 Click **Flow Analysis > Insert > Solid Material**.

2 In the Model Tree, select the **GLASS** component.

3 Under **Solid** expand the list of **Pre-Defined** solid materials and select **Glass** under **Glasses and Minerals**.

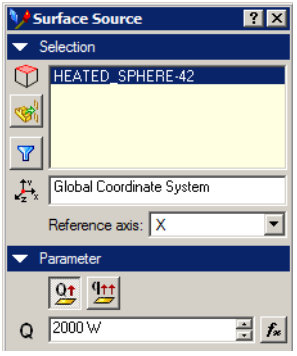
4 Under **Radiation Transparency** select **Transparent**, then select **Thermal only**.

📖 *You can separately specify a solid material transparency to the solar radiation and transparency to thermal radiation from all other sources, including heated bodies. Since there are no sources of solar radiation in the project, we can select **Thermal only** to make the material fully transparent to all radiation in the project.*

5 Click **OK**. FloEFD now treats this solid material and all bodies it is applied to as fully transparent to the thermal radiation.

Specifying Heat Source and Goals

Specify the surface heat source of the heat generation rate at the sphere surface:



1 Click **Flow Analysis > Insert > Surface Source**.

2 In the Model Tree select the **HEATED_SPHERE** component.

3 Select **Heat Generation Rate** as the source type and on the **Settings** tab set its value to 2000 W.

Specify surface goals of the maximum, average, and minimum temperatures at the **HEATED_SPHERE** surface and the blackbody surface of **SCREEN**.
In addition, specify the volume goal of the **HEATED_SPHERE** average temperature. (In all

Goals
SG Max T of Heated Sphere
SG Average T of Heated Sphere
SG Min T of Heated Sphere
SG Max T of Screen
SG Min T of Screen
SG Average T of Screen
VG Average T of Heated Sphere

cases you should select **Temperature (Solid)** as the goal parameter). You can rename the goals as shown to make it easier to monitor them during the calculation.

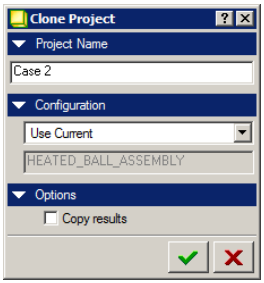
Save the model and run the calculation.

If you take a look at the goals convergence, you can see that the sphere's temperature at the start of the calculation is high. This happens because the initial sphere's temperature (293.2 K) is too low to take away by radiation the heat produced by the 2000 W heat source. To illustrate this better, in cases number 2 and 3 we will increase the initial temperature of the heated sphere to 1000 K, thus providing the greater amount of heat being lost by the sphere starting from the very beginning of the calculation.

Case 2: All reflector surfaces are blackbody

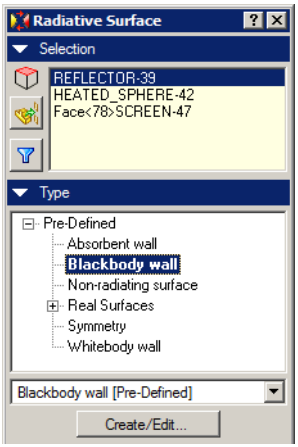
In contrast to the Case 1, in this case the reflector inner surface is blackbody and all other surfaces of the reflector are also blackbody.

Create a new **Case 2** project by cloning the current **Case 1** project.



Changing the Radiative Surface Condition

1. Delete the **Whitebody Wall** condition.
2. Right-click the **Blackbody Walls** item and select **Edit Definition**.
3. Click the **REFLECTOR** item in the Model tree. The component is added to the list.
4. Click **OK** .



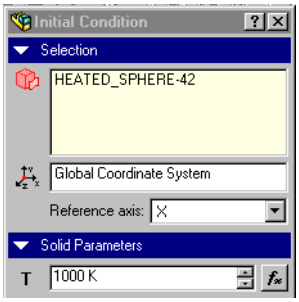
Specifying Goals

Specify the additional surface goals of the maximum, average, and minimum temperature of solid for the **REFLECTOR** inner and outer surfaces.

Specifying Initial Condition in Solid

Specify the initial temperature of the heated sphere of 1000 K using **Initial Condition**.

Save the model.

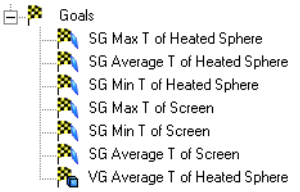


Case 3: The reflector is removed

In contrast to **Case 1** and **Case 2**, the reflector is removed in **Case 3**.

Create a new **Case 3** project by cloning the current **Case 2** project.

1. Edit definition of the **Blackbody Walls** condition: under **Selection** remove from the list all faces belonging to **Reflector**. To delete a face/component from the list of **Faces to Apply the Radiative Surface**, select the face/component and press the **Delete** key.
2. Delete the surface goals related to reflector.
3. Disable the **REFLECTOR** component in the **Component Control** dialog box.

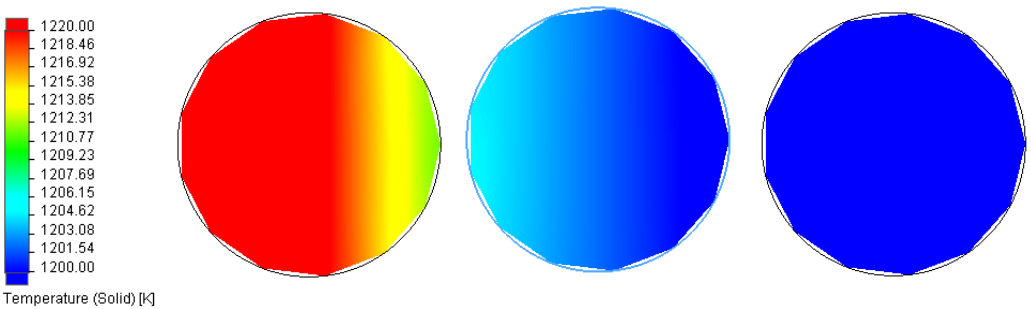

Goals
SG Max T of Heated Sphere
SG Average T of Heated Sphere
SG Min T of Heated Sphere
SG Max T of Screen
SG Min T of Screen
SG Average T of Screen
VG Average T of Heated Sphere

Using **Batch Run**, calculate the cases 2 and 3.

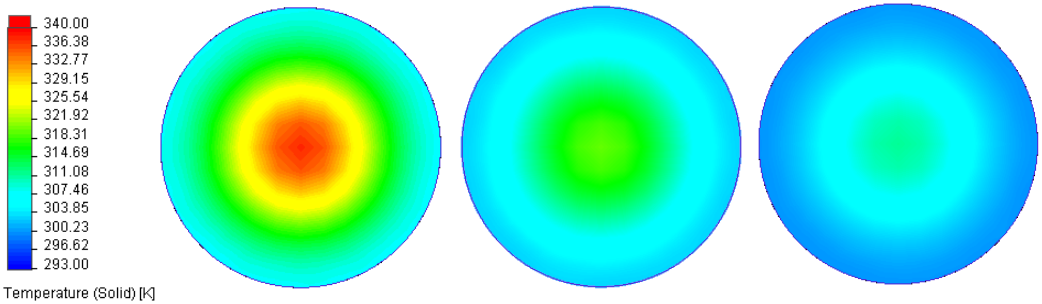
Results

In Case 1, due to the heat returned by the reflector, the ball surface facing the reflector is hotter than the ball surface facing the screen (see pictures below). Therefore, the screen temperature in Case 1 is higher than in the other cases.

In Case 2, radiation coming from the ball to the reflector heats up the reflector and heat is radiated from the reflector outer surface to ambient, therefore being lost from the system. Since less heat is returned to the ball by the radiation from the reflector, the ball temperature is lower, although it is distributed over the ball in the same manner as in Case 1. Less heat is also coming from the reflector to the screen. As a result, the screen temperature is lower than in Case 1.
Since the reflector is removed in Case 3, there is no noticeable heat radiated back to the ball. The ball temperature is lower than in Case 2 and mostly uniform (the non-uniformity is lower than 1 K). Since in the absence of reflector the screen is only exposed to the radiation from the side of the ball facing the screen, the screen temperature is the lowest among all the cases.



The ball temperature distribution (ASM_FRONT plane cross-section) in Case 1 (left), Case 2 (center) and Case 3 (right) in the range from 1200 to 1220 K (the reflector is arranged at the left).



The screen temperature distribution (surface plot of solid temperature) in Case 1 (left), Case 2 (center) and Case 3 (right) in the range from 295 to 340 K.

Parameter		Case 1	Case 2	Case 3
The ball's temperature, K	Maximum	1251,56	1231,28	1222,93
	Average	1228,66	1211,29	1204,06
	Minimum	1210,7	1199,37	1194,32
The screen's temperature, K	Maximum	340,85	322,18	311,91
	Average	317,82	308,97	303,59
	Minimum	307,64	303,18	299,85

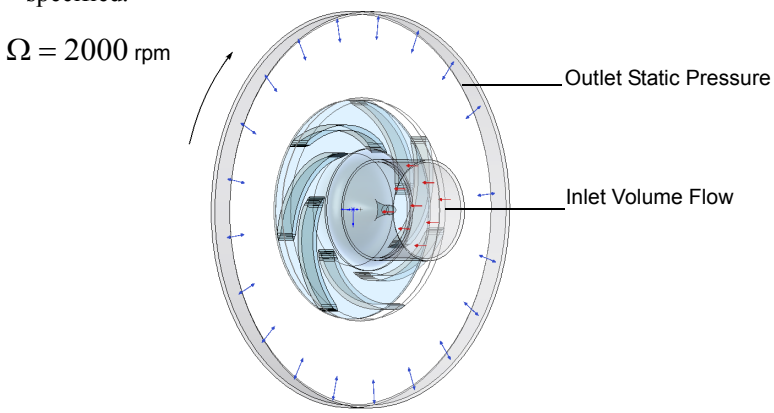
C5

Rotating Impeller

Problem Statement

Let us consider the air flow through a centrifugal pump having a rotating impeller (see below). This pump has a stationary axial inlet (an eye), a pipe section of 92 mm radius with a central body of circular arc contour, which turns the flow by 90^o from the axial direction. At the inlet's exit the radial air flow is sucked by a rotating impeller, which has seven untwisted constant-thickness backswept blades with wedge-shape leading and trailing edges. Each blade is cambered from 65^o at the impeller inlet of 120 mm radius to 70^o at the impeller exit of 210 mm radius, both with respect to the radial direction. These blades are confined between the impeller shrouding disks rotating with the same (as the blades) angular velocity of 2000 rpm. Downstream of the impeller the air enters a stationary (non-rotating) radial diffuser.

To complete the problem statement, let us specify the following inlet and outlet boundary conditions: inlet air of 0.3 m^3/s volume flow rate having uniform velocity profile with vectors parallel to the pump's axis; at the radial-directed outlet a static pressure of 1 atm is specified.



The centrifugal pump with a rotating impeller.

Opening the Model

Copy the **C5 - Rotating Impeller** folder into your working directory. Open the **pump.asm** assembly.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **pump.asm** assembly located in the **C5 - Rotating Impeller\Ready To Run** folder and run the project.*

Creating a FloEFD Project

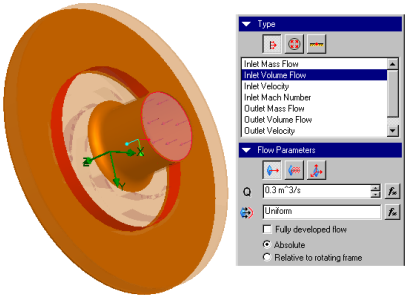
Using the **Wizard** create a new project as follows:

Project name	*Impeller Efficiency*
Configuration	*Use current*
Unit system	**SI**
Analysis type	**Internal,** **Exclude cavities without flow conditions**
Physical features	**Rotation:** *Type:* **Global rotating**, *Rotation axis:* **Z** *axis of* **Global Coordinate system**, **Angular velocity** = `2000 RPM` *(209.43951 rad/s)*
Default fluid	**Air**
Wall Conditions	*Adiabatic wall, default smooth walls*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Set the* **Result resolution level** *to* `4`*;* **Minimum gap size** = `0.04 m`, **Minimum wall thickness** = `0.01 m`, *other options are default*

Specifying Boundary Conditions

Specify the boundary conditions for inlet and outlet flows as shown in the tables below:

Type	**Inlet Volume Flow**
Name	**Inlet Volume Flow 1**
Faces to apply	the inner face of the **INLET_LID**
Parameters: **Volume Flow Rate** of `0.3 m^3/s`, with the **Uniform** profile, in the absolute frame of reference (the **Absolute** option is selected)	

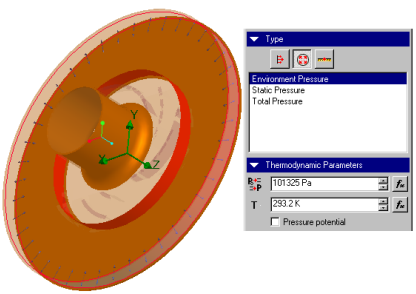


📖 ***Relative to rotating frame***. *When the* ***Relative to rotating frame*** *option is selected, the specified velocity (Mach number) is assumed to be relative to the rotating reference frame (Vr):*

$$V_{specified} = V_r = V_{abs} - \omega \times r$$

Here, r is the distance from the rotation axis and ω *is the angular velocity of the rotating frame. The mass or volume flow rate specified in the rotating reference frame (the Relative to rotating frame option is selected) will be the same in the absolute (non-rotating) frame of reference if the tangential velocity component is perpendicular to the opening's normal, thus not influencing the mass (volume) flow rate value, e.g. when the opening's normal coincides with the rotation axis.*

Type	**Environment Pressure**
Name	**Environment Pressure 1**
Faces to apply	the inner face of the **OUTLET_LID**
Thermodynamic Parameters: Default values (101325 Pa and 293.2 K) in the absolute frame of reference (the **Pressure potentia**l option is not selected)	



📖 ***Pressure potential***. *If you enable a rotating reference frame, you can select the* ***Pressure potential*** *check box. When the Pressure potential check box is selected, the specified static pressure is assumed to be equal to the rotating frame pressure* (P_r) *and may be calculated using following parameters: absolute pressure, density, angular velocity and radius:*

$$P_{specified} = P_r = P_{abs} - \frac{1}{2}\rho\omega^2 \cdot r^2$$

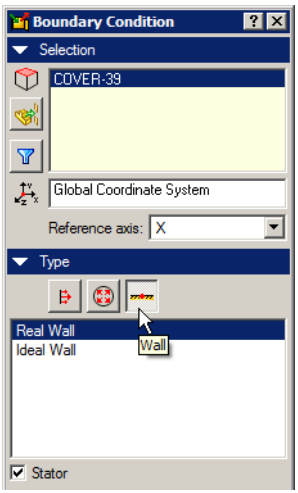
*When the **Pressure potential** check box is unchecked, the specified static pressure is assumed to be a pressure in terms of the absolute frame of reference (P_{abs}).*

When you specify a rotating reference frame, it is assumed that all model walls are rotated with the reference frame's angular velocity unless you set a specific wall to be stationary. To specify a non-rotating wall, the Stator moving wall boundary condition can be applied to this wall. Specifying the stator boundary condition is the same as specifying the zero velocity of this wall in the absolute (non-rotating) frame of reference. Note that stator face must be axisymmetric with respect to the rotation axis.

Specifying Stationary Walls

We will specify the stator condition at the corresponding walls of the pump's cover.

1. In the Model tree, select the **COVER** component.
2. Click **Flow Analysis > Insert > Boundary Condition**.
3. Click **Wall** and keep the default **Real Wall** condition type.
4. Select **Stator**.
5. Click **OK** and rename the new `Real Wall 1` condition to `Stator Walls`.



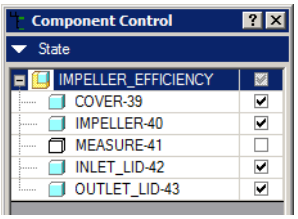
On Calculating the Impeller's Efficiency

Engineers dealing with pump equipment are interested in the pump efficiency. For the pump under consideration the efficiency (η) can be calculated in the following way (F.M.White "Fluid Mechanics", 3rd edition, 1994):

$$\eta = \left| \frac{\left(P_{outlet} - P_{inlet}\right) \cdot Q}{\Omega \cdot M} \right|$$

where P_{inlet} is the static pressure at the pump's inlet, P_{outlet} is the bulk-average static pressures at the impeller's outlet (Pa), Q is the volume flow rate (m^3/s), Ω is the impeller rotation angular velocity (rad/s), and M is the impeller torque (N·m). To obtain P_{outlet}, an auxiliary **MEASURE** component was placed where the flow exits the impeller.

The **MEASURE** component is only used for the pressure measurement (the corresponding goal will be specified at the inner face of the **MEASURE** thin ring), thus it should be disabled in the **Component Control** dialog box.



1 Click **Flow Analysis > Component Control**.

*In Creo Parametric click **Flow Analysis > Component Control**.*

2 Deselect the **MEASURE** component.

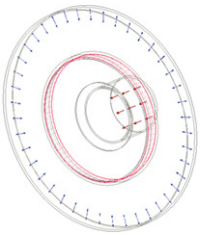
3 Click **OK** to close the dialog.

Specifying Project Goals

First, since the pressure and volume flow rate boundary condition are specified, it makes sense to set the mass flow rate surface goal at the pump's inlet and outlet to inspect the mass balance as an additional criterion for converging the calculation.

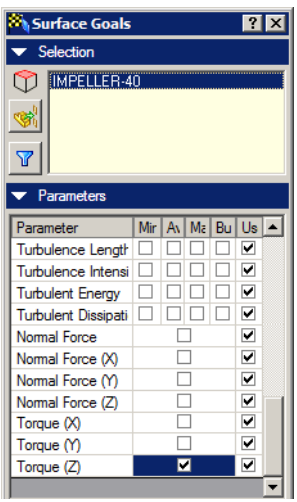
GOAL TYPE	GOAL PARAMETER	FACE
Surface Goal	Mass Flow Rate	The inner face of the **INLET_LID**
Surface Goal	Mass Flow Rate	The inner face of the **OUTLET_LID**

Next, specify the goals that are necessary for calculating the impeller's efficiency:

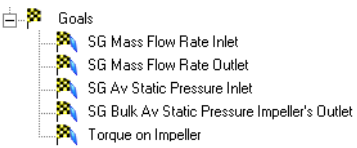
GOAL TYPE	GOAL PARAMETER	FACE
Surface Goal	Av Static Pressure	The inner face of the **INLET_LID**
Surface Goal	Bulk Av Static Pressure	The inner face of the **MEASURE** ring at the impeller's outlet. 
Surface Goal	Torque (Z)	All **impeller** faces in contact with air (see details below).

1 Select the **IMPELLER** component by clicking on it in the geometry area or in the Model tree.

2 Click **Flow Analysis > Insert > Surface Goals**. The **IMPELLER** component appears in the **Faces to Apply the Surface Goal** list.



Rename the created goals as shown below:


Goals
SG Mass Flow Rate Inlet
SG Mass Flow Rate Outlet
SG Av Static Pressure Inlet
SG Bulk Av Static Pressure Impeller's Outlet
Torque on Impeller

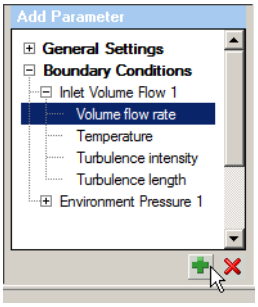
Finally, specify the following Equation goals:

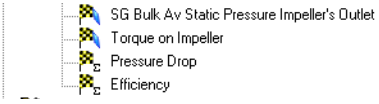
GOAL NAME	FORMULA	DIMENSIONALITY
Pressure Drop	{SG Av Static Pressure Inlet}-{SG Bulk Av Static Pressure Impeller's Outlet}	Pressure & stress
Efficiency	{Pressure Drop}*{Inlet Volume Flow 1:Volume flow rate:3.000e-001}/209.44/{Torque on Impeller}	No unit

To add inlet volume flow value to the equation goal's expression:

1 On the Equation goal pane click **Add Parameter** .

2 In the **Add Parameter** list expand the **Boundary Conditions** group, under **Inlet Volume Flow 1** select **Volume flow rate** and click **Add** .

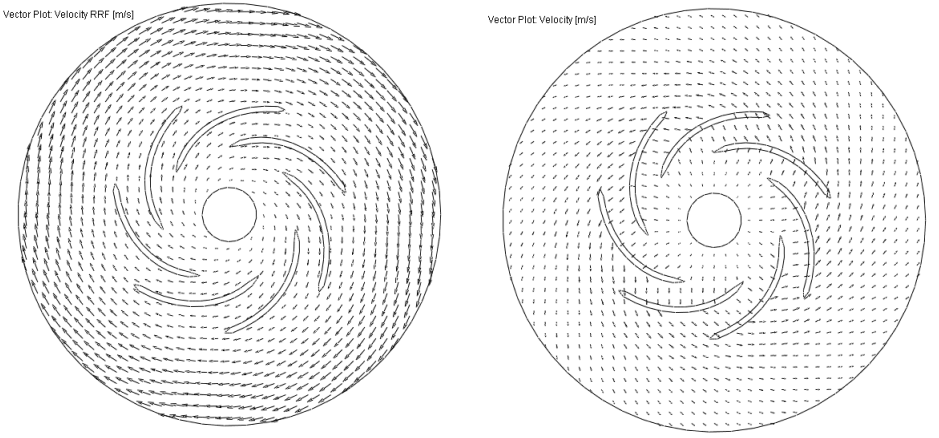




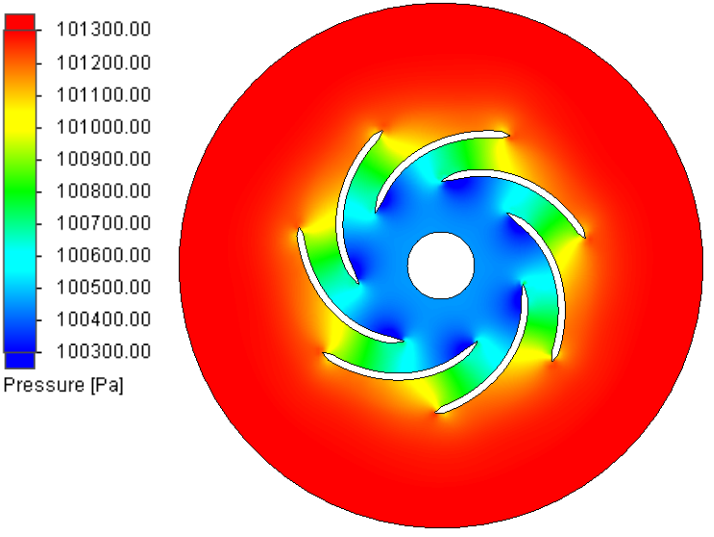
Save the model and run the calculation.

Results

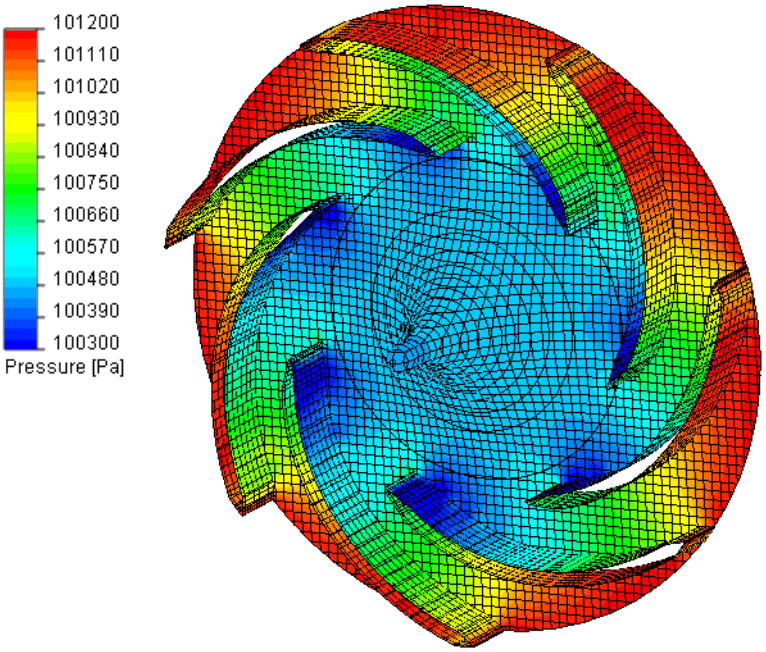
The velocity vectors and static pressure distribution are shown below. To display vectors in the rotating reference frame, select the **Velocity RRF** parameter under the **Vectors** of the **Cut Plot** definition window.



The flow velocity vectors in the frame rotating with the impeller (left) and in the stationary frame (right) at the impeller flow passage midsection (ASM_FRONT cross-section, position *Z = - 0.02 m*, vector spacing = 0.02 m, arrow size = 0.03 m).



The flow static pressure at the impeller flow passage midsection.



The flow pressure distribution over the impeller surface.

For the impeller under consideration the obtained efficiency is about 0.75.

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value
Efficiency	[ ]	0,75704682	0,757049362	0,756981666	0,757112269

C6

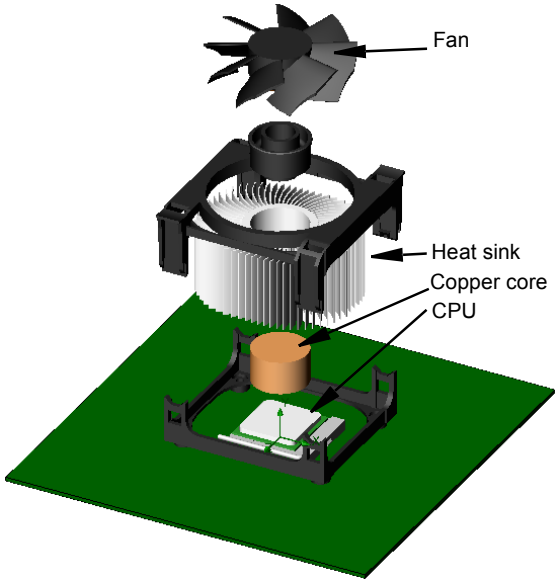
CPU Cooler

Problem Statement

Let us consider a CPU cooler consisting of a copper core and an aluminum heat sink with 62 fins. An eight-blade propeller generates a constant flow of air through the heat sink. The CPU is mounted on a socket installed on a PCB. Heat produced by the CPU is transferred through the core to the heat sink and then released into the air flow.

To calculate the problem using FloEFD, it is convenient to use the concept of local rotating regions. In order to simplify the problem statement, we do not consider the thermal interface layer between the processor and the cooler. Also, we neglect the thermal conduction through the processor socket and PCB.

A quantitative measure of the cooler efficiency is the thermal characterization parameter $\Psi_{CA} = (T_C - T_A)/P_D$, where T_c is the temperature of the CPU cover, T_A is the surrounding air temperature, and P_D is the thermal design power (TDP) of the CPU.



An exploded view of the CPU cooler assembly.

Opening the Model

Copy the **C6 - CPU Cooler** folder into your working directory. Open the **CPU_COOLER.ASM** assembly.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **CPU_COOLER.ASM** assembly located in the **C6 - CPU Cooler\Ready To Run** folder and run the project.*

Creating a FloEFD Project

Using the **Wizard** create a new project as follows:

Project name	*CPU Cooler at 4400 RPM*
Configuration	*Use current*
Unit system	**SI**
Analysis type	**External***;* **Exclude cavities without flow conditions***;* **Exclude internal space**
Physical features	**Heat conduction in solids***;* **Rotation:** *Type:* **Local region(s)**
Default fluid	**Gases / Air**
Default solid	**Glasses and Minerals / Insulator**
Wall Conditions	*Default smooth walls*
Initial and Ambient Conditions	**Thermodynamic parameters***:* **Temperature** = 38°C*;* **Solid parameters***:* **Initial solid temperature** = 38°C*; other conditions are default*
Result and Geometry Resolution	*Set the Result resolution level to 5;* *Minimum gap size = 0.001 m,* *other options are default*

Adjusting the Computational Domain Size

Specify the computational domain size as follows:

X max = 0.095 m	Y max = 0.1123 m	Z max = 0.095 m
X min = -0.095 m	Y min = 0.0005 m	Z min = -0.095 m

Specifying the Rotating Region

The **Rotating region** is used to calculate flow through rotating components of model (fans, impellers, mixers, etc.) surrounded by non-rotating bodies and components, when a global rotating reference frame cannot be employed. For example, local rotating regions can be used in analysis of the fluid flow in the model including several components rotating over different axes and/or at different speeds or if the computational domain has a non-axisymmetrical (with respect to a rotating component) outer solid/fluid interface. Each rotating solid component is surrounded by an axisymmetrical rotating region which has its own coordinate system rotating together with the component.

A rotating region is defined by an additional component of the model. This additional component must meet the following requirements:

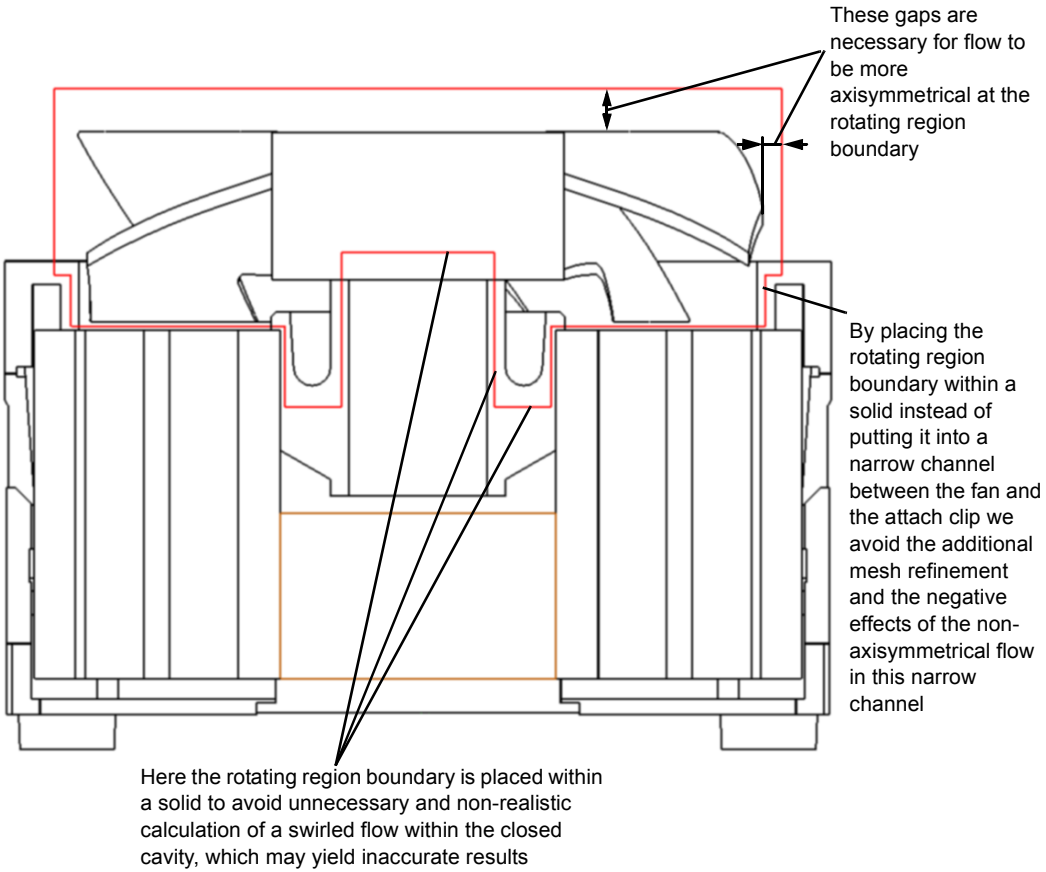
- the rotating component must be fully enclosed by it,
- it must be axisymmetric (with respect to the rotating component's rotation axis),
- its boundaries with other fluid and solid regions must be axisymmetrical too, since the boundaries are sliced into rings of equal width and the flow parameters' values transferred as boundary conditions from the adjacent fluid regions are circumferentially averaged over each of these rings,
- the components defining different rotating regions must not intersect.

Specify the rotating region as follows:

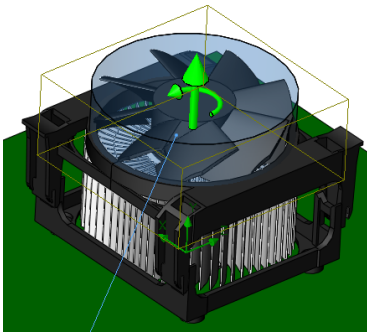
1. Click **Flow Analysis > Insert > Rotating Region**.
2. In the Model tree, select **ROTATION_REGION** component. Note that the **Disable solid components** check box is automatically selected to treat the Rotating Region as a fluid region.

📖 *A component to apply a rotating region must be a body of revolution whose axis of revolution is coincident with the rotation axis. This component must be disabled in the* **Component Control**. *When specifying the rotating region, make sure that its boundaries do not coincide with the boundaries of other surrounding solid components, because the mesh will not resolve this region. However these components may intersect in some way, but in this case the surrounding components must be also symmetrical relatively to the axis of revolution. Since the flow on the boundary of the rotating region must be axisymmetrical as well, we must provide a reasonable gap between the rotating region boundary and the outer edges of the propeller blades in*

order to minimize the influence of local non-axisymmetrical perturbations. Due to the same reason, it is preferable to put the rotating region boundary inside the solid bodies whenever possible, rather than putting them in the narrow flow passages. Also, the supposed direction of the flow at the rotating region boundary should be taken into account when defining the shape of the rotating region. You should choose such shape of the rotating region that the flow direction will be as much perpendicular to the rotating region boundary as possible. The picture below provides an additional insight into how the rotating region shape was adapted to the actual geometry of the CPU cooler in this tutorial example (the rotation region boundary is denoted by red).


These gaps are necessary for flow to be more axisymmetrical at the rotating region boundary
By placing the rotating region boundary within a solid instead of putting it into a narrow channel between the fan and the attach clip we avoid the additional mesh refinement and the negative effects of the non-axisymmetrical flow in this narrow channel
Here the rotating region boundary is placed within a solid to avoid unnecessary and non-realistic calculation of a swirled flow within the closed cavity, which may yield inaccurate results

3 Under **Parameter**, set the **Angular Velocity** to 4400 RPM. If default direction of the rotation is opposite to the desired, then specify -4400.



🕮 *During the definition of a rotation region, heavy green arrows denoting the rotation axis and the default direction of rotation speed can be seen in the graphics area. Since we want to define the rotation in the direction opposite to the default, we specify negative value of the angular velocity.*

4 Click **OK**.

When you specify a rotating region, it is assumed that all model walls within this region rotate with the region's angular velocity unless you set a specific wall to be stationary. To specify a non-rotating wall, the Stator real wall boundary condition should be applied to the wall. Specifying the stator boundary condition is the same as specifying the zero velocity of this wall in the absolute (non-rotating) frame of reference. Note that the stator face (or a part of the face that is located inside the rotating region in the case when the given face intersects with the rotating region boundary) must be axisymmetric with respect to the rotation axis.

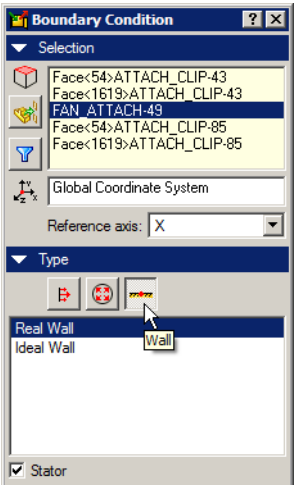
Specifying Stationary Walls

We will specify the stator condition at the appropriate walls of the fan attach and the attachment clip. To easily select the necessary faces, hide the **FAN** and **ROTATING_REGION** components.

1 Click **Flow Analysis > Insert > Boundary Condition**.

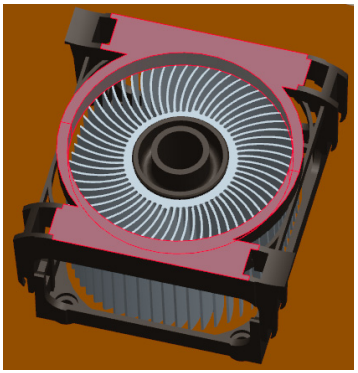
2 Under **Type**, select **Wall**. Keep the default **Real Wall** condition type and select **Stator** option.

3 In the Model tree select the **FAN_ATTACH** component.



4 Holding the CTRL key, select the two inner circular side faces and two top faces of **ATTACH_CLIP** as shown.

5 Click **OK**.



Specifying Solid Materials

Specify the solid materials for the project as follows:

a) the **CPU** and the **HEAT_SINK** are made of Aluminum (Pre-Defined/Metals);
b) the **COPPER_CORE**, naturally, is made of Copper (Pre-Defined/Metals);
c) all other parts are made of default Insulator.

Specifying Heat Source

Specify the volume source with the heat generation rate of 75 W in the **CPU** component.

Specifying Initial Mesh Settings

To resolve the complex geometry of the fan and heat sink better, let us define six additional control planes and specify the proper **Ratio**s for the intervals between them to make the mesh denser in the central region containing the complex geometry and coarser near the computational domain's boundaries.

1 Click **Flow Analysis > Initial Mesh**.

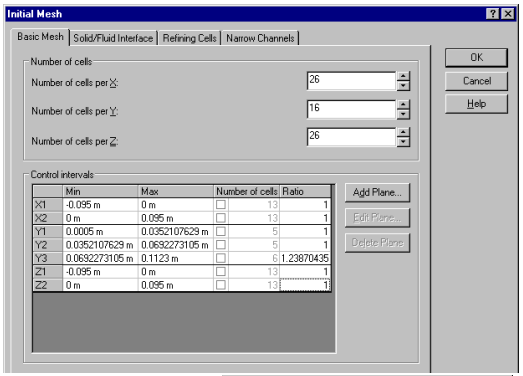
*In Creo Parametric click **Flow Analysis > Mesh > Initial Mesh**.*

2 Clear the **Automatic settings** check box.

3 Click **Reset** and then in the **Automatic Initial Mesh** window click **OK**.

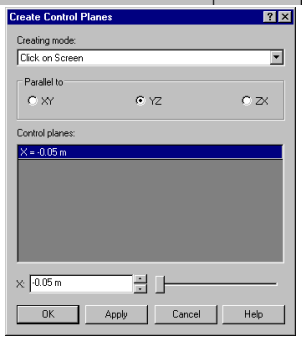
4 On the **Basic Mesh** tab, under **Control intervals** select either **Max** of **X1** interval or **Min** of **X2** interval and click **Delete Plane**.

5 Click **Add plane**.



6 In the **Create Control Planes** window make sure that **Creating mode** is set to **Click on screen** and **Parallel to** is set to **YZ**, click anywhere in the geometry area and enter manually -0.05 as a new value for **X**. Click **OK** to return to the **Initial Mesh** window.

7 Following the same procedure, add one more plane at X = 0.05.

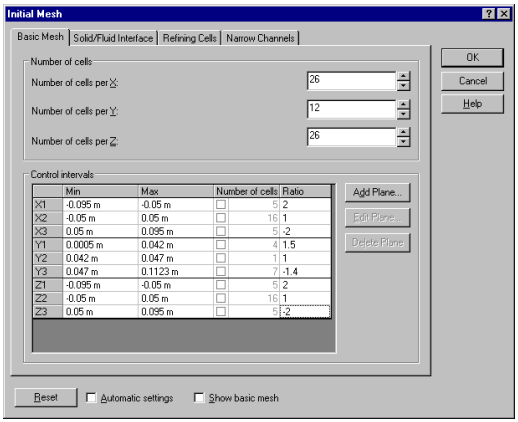


🕮 *By default, FloEFD creates six control planes on the computational domain boundaries and a number of planes inside it. We now want to tune the set of control planes to our needs by removing the default planes inside the computational domain and adding new ones.*

8 Click the **Ratio** cell of the **X1** interval and enter the value of 2. In the same manner enter the values 1 and -2 for the intervals **X2** and **X3**.

🕮 ***Ratio*** *is the ratio of cell sizes on the given interval. The cell sizes are changed gradually along the selected direction so that the proportion between the first and the last cells of this interval is close (but not necessarily equal) to the entered value of the* ***Ratio****. Negative values of the ratio correspond to the reverse order of cell size increase.*

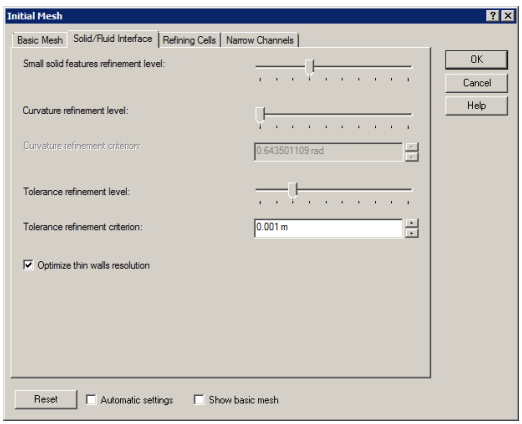
9 Delete any existing inner control planes perpendicular to Y and add new planes at Y = 0.042 m and Y = 0.047 m. Specify the **Ratio** values for **Y1**, **Y2** and **Y3** intervals as 1.5, 1 and -1.4, respectively.



10 Delete the existing inner control plane perpendicular to Z and add new planes at Z = -0.05 m and Z = 0.05 m. Specify the **Ratio** values for **Z1**, **Z2** and **Z3** intervals as 2, 1 and -2, respectively.

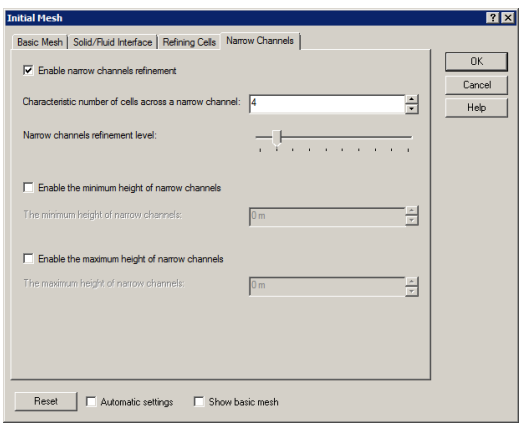
11 Check that the **Numbers of cells per X**, **Y** and **Z** are 26, 12 and 26, respectively. If the numbers are different, please correct them manually.

12 To avoid the unnecessary mesh refinement at the edges of the heatsink fins, go to the **Solid/fluid Interface** tab and set **Small solid features refinement level** to 3, **Tolerance refinement level** to 2, and **Tolerance refinement criterion** to 0.001 m, while leaving other options default.



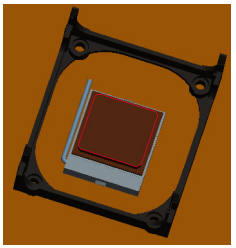
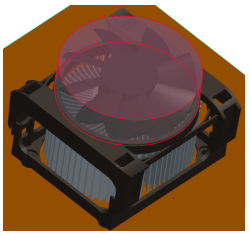
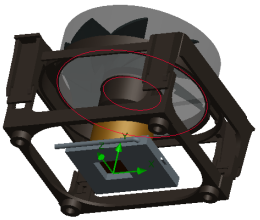
13 Go to the **Narrow Channels** tab and set **Characteristic number of cells across a narrow channel** to 4 and **Narrow channels refinement level** to 1, leaving default values for other options. This will prevent the unnecessary mesh refinement in the narrow channels between heatsink fins.

14 Click **OK**.



Specifying Project Goals

Specify surface goals for maximum temperature on the CPU cover and mass flow rate for the flows entering the rotating region and exiting from it. To select the necessary faces, you will probably need to hide temporarily some components of the assembly.

GOAL TYPE	GOAL VALUE	FACE
Surface Goal	Max Temperature (Solid)	Top face of the CPU cover. To set this goal you may need to hide the **HEAT_SINK** and **COPPER_CORE** components. 
Surface Goal	Mass Flow Rate	Top and side surfaces of the **Rotation Region** component. 
Surface Goal	Mass Flow Rate	Bottom face of the **Rotation Region** component. To set this goal you may need to hide the **PCB** component. 
Equation goal	({SG Mass Flow Rate 1}+{SG Mass Flow Rate 2})/{SG Mass Flow Rate 1}	The disbalance of the inlet and outlet mass flow rates. We are using the "+" operand since the inlet and outlet mass flow rate values have opposite signs. Select **No unit** for **Dimensionality.**

To calculate the thermal characterization parameter we will need the temperature of the center of the CPU cover. To get more accurate value of the parameter we will specify a separate point goal.

1 Click **Flow Analysis > Insert > Point Goals.**

*In Creo Parametric click **Flow Analysis > Insert > Goals > Point Goals**.*

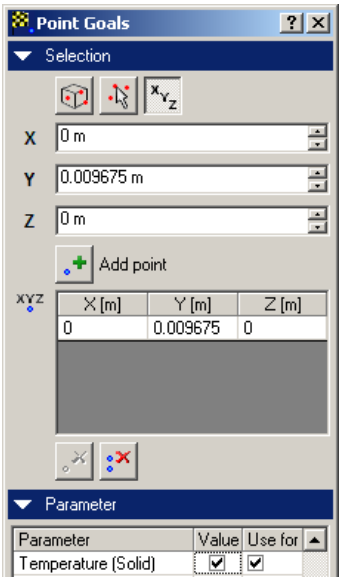
2 Click **Point Coordinates** .

3 Enter the coordinates of the point: **X** = 0 m, **Y** = 0.009675 m, **Z** = 0 m.

4 Click **Add Point** . In the **Parameter** table, select the **Value** check box in the **Temperature (Solid)** row.

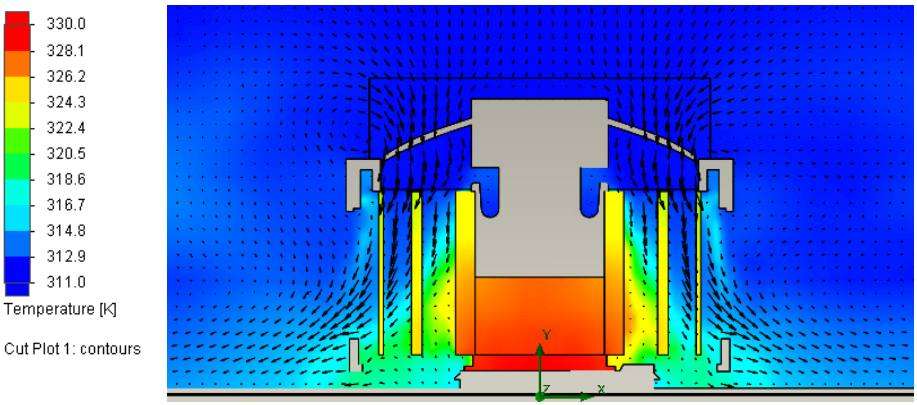
5 Click **OK**.

Save the model and run the calculation.

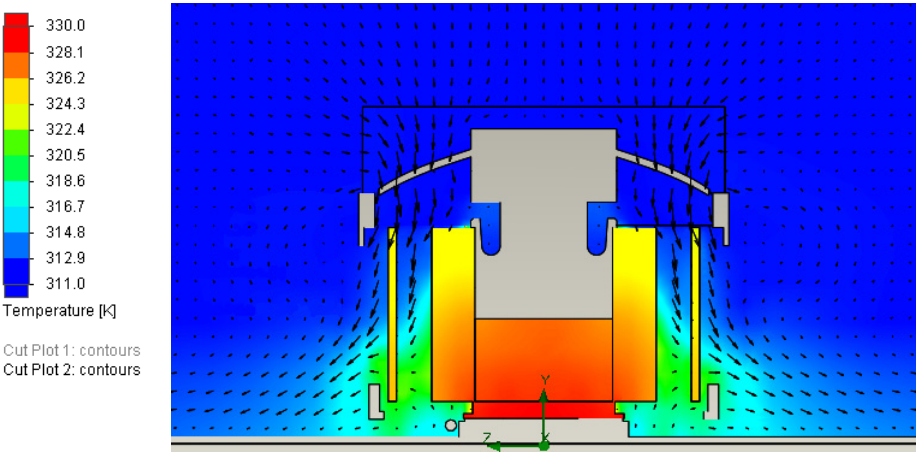


Results

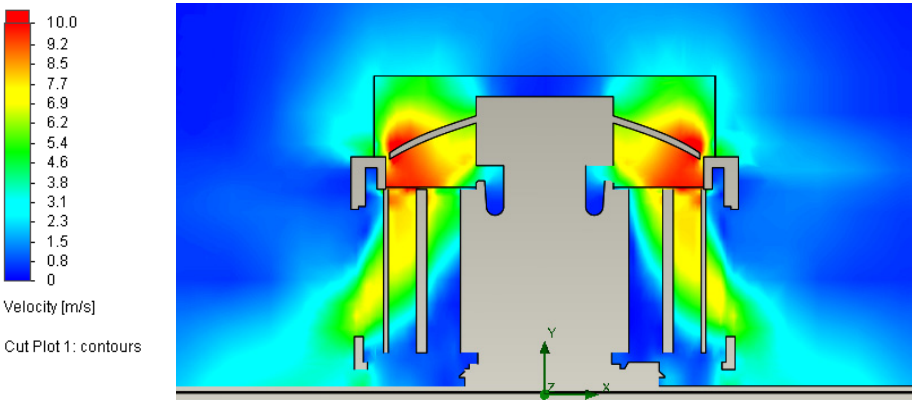
Use the goal plot tool to obtain the value of the temperature of the center of the CPU cover. Now we can calculate the thermal characterization parameter of the heat sink: $\Psi_{CA} = (T_C - T_A)/P_D = (330\text{-}311.15)/75 = 0.25$ °C/W. The second most important characteristic of the CPU Cooler is the velocity of the flow above PCB. We can assess the value of this parameter as well as the distribution of the temperature by looking at the cut plots made in the **ASM_FRONT** and **ASM_RIGHT** planes (see below).



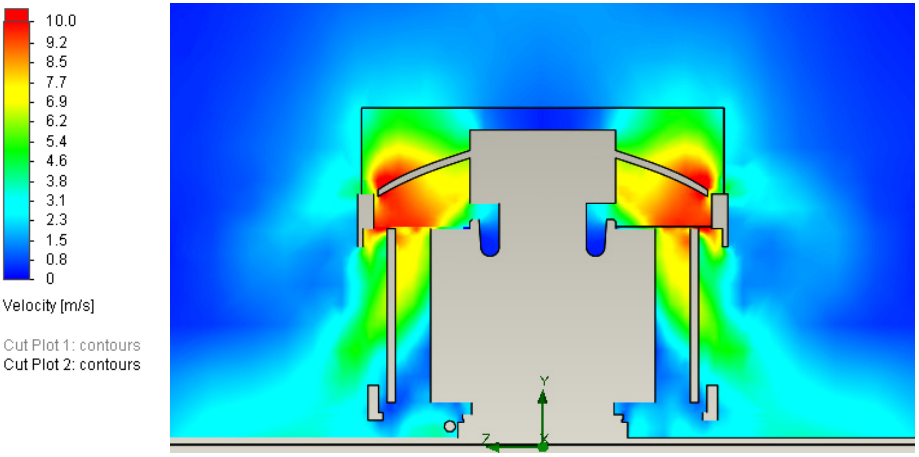
Temperature field and velocity vectors distribution
(ASM_FRONT plane, no offset, vector spacing = 0.003 m, arrow size = 0.015 m).



Temperature field and velocity vectors distribution
(ASM_RIGHT plane, no offset, vector spacing = 0.003 m, arrow size = 0.015 m).



Velocity distribution as a contour plot (ASM_FRONT plane, no offset).



Velocity distribution as a contour plot (ASM_RIGHT plane, no offset).

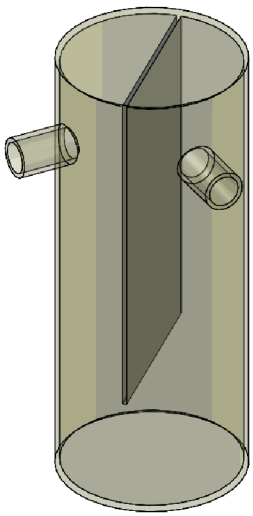
C7

Oil Catch Can

Problem Statement

Here we consider the motion of motor oil droplets in the air flow inside the oil catch can installed in the car. The presence of the droplets in this flow is caused by the rotating crankshaft that churns up oil inside the crankcase. As oil catch can traps these droplets, it eliminates the possibility of oil suction into the engine and its subsequent combustion with fuel and oxidizer (air) that produces a lot of smoke in the exhaust.

For this tutorial we consider the geometry of oil catch can shown on the picture right. The dividing wall is placed so that most of the droplets entering through the inlet nipple along with the air flow collide to it. Once the collision occurs, the oil droplet adheres to the wall and then trickles down. However some particular smaller-sized droplets may evade collision with the wall due to their small inertia and escape the can through the outlet nipple.



The objective of the simulation is to estimate the probability of trapping oil droplets in the oil catch can considering the following droplet sizes: 8, 13 and 18 μm. Quantitatively, we can calculate this probability value for each individual droplet size with the following expression: $P = m_{outlet} / m_{inlet}$,

where m_{inlet}, m_{outlet} is the mass flow rate of oil droplets in the inlet and in the outlet correspondingly. The value of m_{inlet} is set to equal to 0.5% of air mass flow rate.

We assume that oil droplets do not influence the air flow because of their small size and mass (~10^{-13} kg). Therefore, we also neglect the impact of oil accumulation on the flow inside the oil catch can.

Opening the Model

Copy the **C7 - Oil Catch Can** folder into your working directory. Open the **OIL_CATCH_CAN.*asm*** assembly.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **OIL_CATCH_CAN.asm** assembly located in the **C7 - Oil Catch Can\Ready To Run** folder and run the project.*

Creating a FloEFD Project

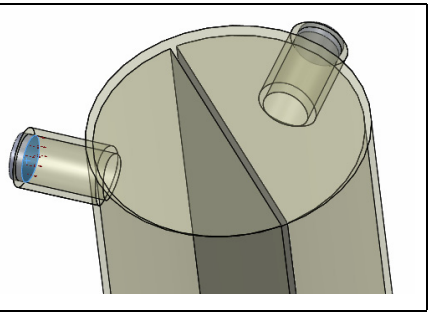
Using the **Wizard** create a new project as follows:

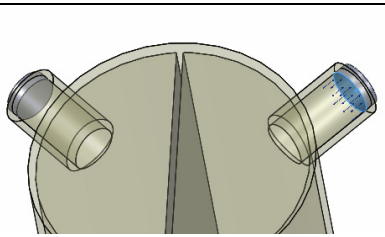
Project name	*Oil particles*
Configuration	*Use current*
Unit system	**SI**
Analysis type	**Internal**
Default fluid	**Gases / Air**
Wall Conditions	*Adiabatic wall with zero roughness*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Default Result resolution level of 3; other options are default*

Specifying Boundary Conditions

Specify the boundary conditions for inlet and outlet flows as shown in the tables below:

Type	**Inlet Volume Flow**
Name	**Inlet Air Volume Flow**
Faces to apply	the inner face of the **INLET_LID**
Parameters: **Volume Flow Rate** of `100 l/min (0.00167 m^3/s)`	



Type	**Static Pressure**	
Name	**Outlet Static Pressure**	
Faces to apply	the inner face of **OUTLET_LID**	
Thermodynamic Parameters: Default values (101325 Pa and 293,2 K)		

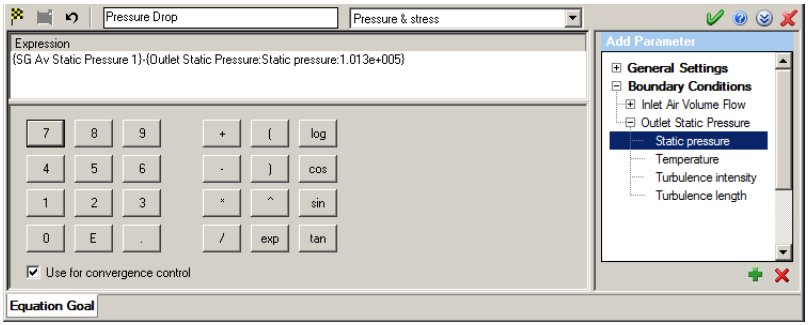
Specifying Project Goals

1. In the **Analysis tree**, select the **Inlet Air Volume Flow** boundary condition
2. Click **Flow Analysis > Insert > Surface Goals**.
3. Under **Parameter**, select **Av Static Pressure**.
4. Click **OK** . This goal will be an intermediate one to calculate pressure drop through the oil catch can.
5. Click **Flow Analysis > Insert > Equation Goal**.
6. On the pane in the bottom of the screen click **Add Goal** .
7. From the **Add Goal** list select the **SG Av Static Pressure 1** goal and click **Add** . It will appear in the **Expression** box.

*In Creo Parametric click **Flow Analysis > Insert > Goals > Equation Goal**.*

8. Click the minus "-" button in the calculator.
9. On the pane in the bottom of the screen click **Add Parameter** .
10. In the **Add Parameter** list expand the **Boundary Conditions** group.
11. Under the **Boundary Conditions** group expand the **Outlet Static Pressure** boundary condition, select **Static Pressure** and click **Add** .
12. Select **Pressure & stress** for **Dimensionality.**

13 Type **Pressure Drop** in the **Goal Name** box.



14 Click **OK**.

15 Click **Flow Analysis > Insert > Global Goals**.

*In Creo Parametric click **Flow Analysis > Insert > Goals > Global Goals**.*

16 Under **Parameter**, select **Torque (Y)**, and click **OK** .

Setting Solution Adaptive Mesh Refinement

With the specified **Result Resolution** value of 3, it may be not sufficient to resolve accurately the regions with large velocity gradients and swirls, which are obviously present here. When analyzing the particles, this may also lead to incorrect predictions of particle trajectories. So, to improve the accuracy of the solution in those regions, it is convenient to perform additional (adaptive) mesh refinement during the calculation.

1 Click **Flow Analysis > Calculation Control Options**.

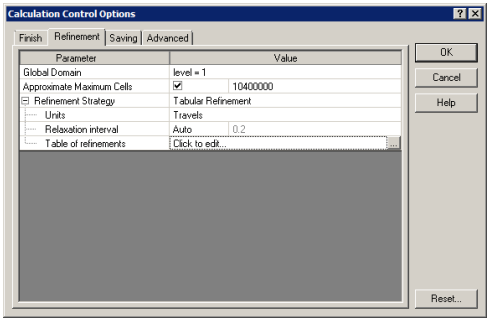
*In Creo Parametric click **Flow Analysis > Project > Calculation Control Options**.*

2 Go to the **Refinement** tab.

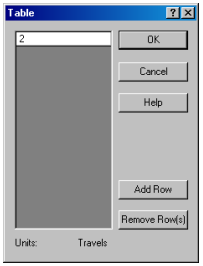
3 Under **Global Domain** specify refinement **level = 1**.

4 Make sure that the value of the **Refinement Strategy** item is set to **Tabular Refinement**.

5 To edit the table of refinements, first expand the **Refinement Strategy** item and make sure that the value of **Units** is set to **Travels**. Then, click the ... button in the **Table of refinements** field.



6 In the opened window, click **Add Row**. A single blank row will appear.



7 Enter the value of 2 in the created row. This means that mesh refinement will occur during the calculation when the value of travels reaches 2.

8 Click **OK**. Go to the **Finish** tab.

9 Under the **Finish Conditions**, make sure that the **Minimum refinement number** is set **On**. Edit its value to 1.

10 Set **Off** the **Maximum travels**.

11 Click **OK**.

Save the model and run the calculation. During the calculation you can preview the velocity field in the **Front Plane** or other plane and see how mesh refinement improves the final solution.

Defining Motor Oil Material

1 Click **Flow Analysis > Tools > Engineering Database**.

2 In the **Database tree**, select **Materials > Liquids > User Defined**.

3 Click **New Item** in the toolbar. The blank **Item Properties** tab appears. Double-click the empty cell to set the corresponding property value.

4 Specify the material properties as shown in the table below:

Name	`Tutorial Motor Oil`
Density	`900 kg/m^3`
Dynamic viscosity	`0.01 Pa*s`
Specific heat	`1900 J/(kg*K)`
Thermal conductivity	`0.2 W/(m*K)`

Studying the Motion of Oil Droplets

1 In the Analysis tree, right-click the **Particle Studies** icon and select **Wizard**.

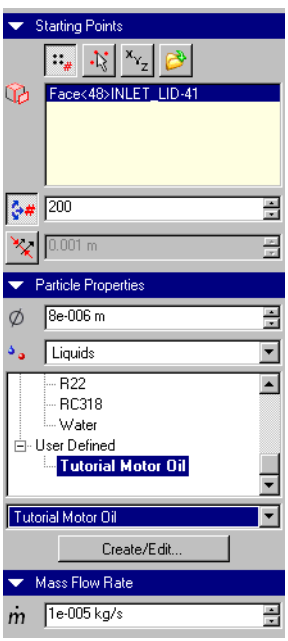
2 Keep the default name for the Particle Study and click **Next**.

3 Click the **Inlet Air Volume Flow** boundary condition, so that the corresponding face appears under the **Starting Points**.

4 Set the **Number of points** to 200.

5 Under the **Particle Properties**, set the **Diameter** equal to `8e-06 m` and change the **Material** to the created **Tutorial Motor Oil** (**Materials / Liquids / User Defined**).

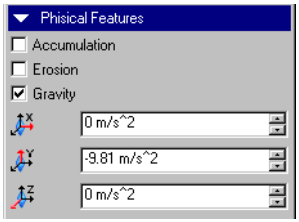
6 Change the **Mass flow rate** value to `1e-05 kg/s`. This value is obtained once we take the 0.5% of inlet air mass flow rate (the product of volume flow rate and density) in accordance with the problem statement.



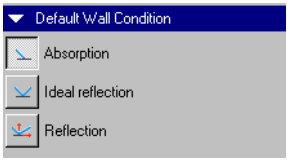
📖 *The value set for the **Number of points** reflects the number of different possible trajectories of the considered particles used for tracing. Obviously the larger this value is, the more accurate information about possible particle trajectories can be obtained. As a result, you can obtain a more detailed picture of the particles' distribution in the considered domain and, if necessary, calculate their mass flow rate in the outlet with a higher precision.*

7 Click **Next** .

8 Under **Physical Features**, select **Gravity**. Click **Next** .



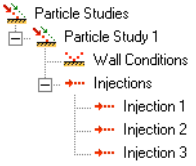
9 Under the **Default Wall Condition**, make sure that **Absorption** is selected. Click **Next** .



📖 *For the **Particle Study**, there are three types of boundary conditions that can be assigned to the walls: **Ideal reflection**, **Absorption** and **Reflection**. The first two indicate perfectly elastic and inelastic collision respectively. In the third type, you have to specify the restitution coefficients that define the ratios of normal and tangential (to the wall) velocity components after and before the collision.*

10 Under **Default Appearance**, set **Draw Trajectories as** **Lines**.

11 Click **OK** . A new **Particle Study 1** item with one sub-item (**Injection 1**) appear in the **Analysis tree**.

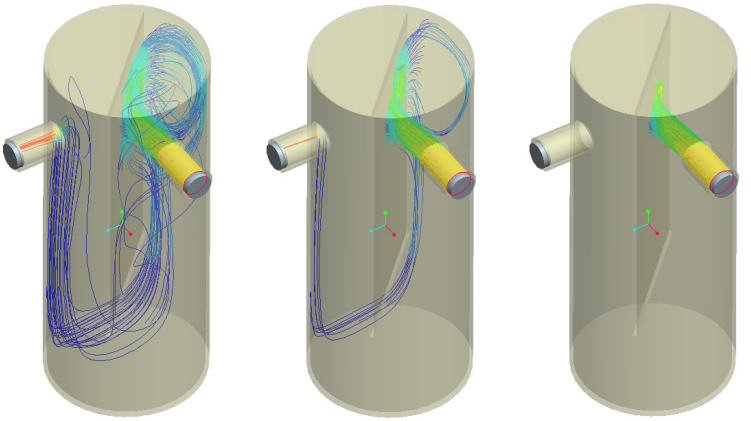


12 Right-click the created **Injection 1** item and select **Clone**. Create this way **Injection 2** and **Injection 3** items.For the **Injection 2** and **Injection 3**, edit the **Diameter** to `1.3e-05 m` and `1.8e-05 m` respectively.

13 Right-click the **Particle Study 1** item and select **Run**.

Results

You can see the trajectories of each droplet size (injection), by right-clicking on the **Injection** of interest and selecting **Show**. The resulting trajectories colored by the **Velocity** parameter are presented below.



8µm droplets 13µm droplets 18µm droplets

For each particular droplet size, we can obtain the precise amount of particles flown out of the Oil Catch Can by evaluating the integral parameter **Number of Particles** on the outlet face using the **Surface Parameters** feature.

With these values, we can conclude that the probability of trapping the 18µm droplets is 100%; 13µm is about 97%; 8µm is about 90%.

D

Examples for HVAC Module

The examples for **HVAC Module** presented below demonstrate how to use capabilities and features of this module to solve real-life Heating, Ventilation, and Air Conditioning problems. This functionality is available for the HVAC module users only.

D1 - 150W Halogen Floodligh

D2 - Hospital Room

D3 - Pollutant Dispersion in the Street Canyon

Examples for HVAC Module:

D1

150W Halogen Floodlight

This feature is available for the HVAC module users only.

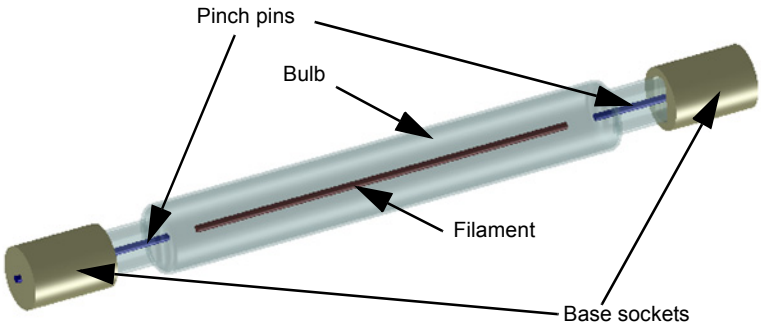
Problem Statement

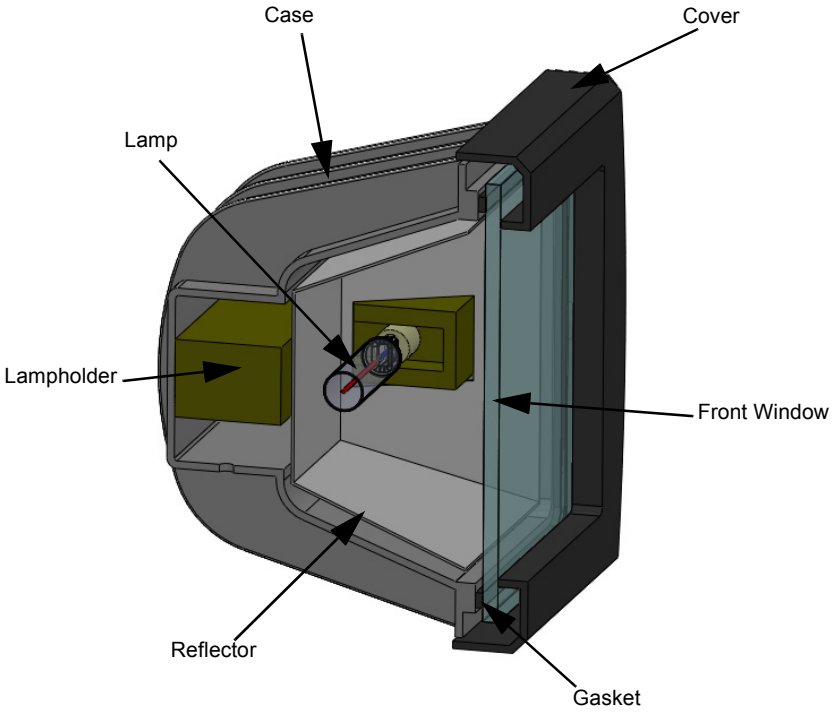
This tutorial demonstrates the capability of FloEFD to simulate heat transfer by convection and radiation, including the radiation absorption in semi-transparent solids and the radiation spectrum. It is shown how to define a project, specify the radiation properties of semi-transparent solid materials, radiation conditions and calculation goals.

Here we consider a halogen floodlight with an aluminum housing, which contains a quartz glass front window, a silicone gasket, an aluminum internal reflector, a ceramic lampholder and a 150W linear halogen lamp.

The linear halogen lamp consists of a quartz glass bulb, a straight-line tungsten filament, molybdenum pinch pins and ceramic base sockets. The lamp is filled with argon at 2 atm and 293.2 K. The lamp operates in typical indoor conditions at the room temperature (~20 °C) and without any forced cooling.

Components of the floodlight and the halogen lamp are shown at the figures below.





In the table below, you can see the typical values of the maximum allowable operating temperatures for some of these components. The objective of the simulation is to ensure that the pinch pins, the lamp bulb and the front glass are not overheated.

Component	**Maximum permissible temperature**
Pinch pins	350 °C
Bulb glass	900 °C

Opening the Model

Copy the **D1 - Halogen Floodlight** folder into your working directory. Open the **Floodlight.asm** assembly. Select the generic instance.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **Floodlight.asm** assembly located in the **D1 - Halogen Floodlight\Ready To Run** folder and run the project.*

Creating a FloEFD Project

Using the **Wizard** create a new project as follows:

Project name	*Floodlight*
Configuration	*Use current*
Unit system	**SI**
Analysis type	**External**
Physical features	**Heat conduction in solids**, **Radiation:** **Radiation model: Discrete Ordinates** **Environment radiation:** **Environment temperature** *of* `293.2 K`**;** **Absorption in solids;** **Spectrum: Number of bands** *is* **2**, **Bands edge 1** *at* `2500 nm;` **Gravity***:* **Y component** *of* `-9.81 m/s^2`
Default fluid	**Gases / Air** *(***Default fluid***)* **Gases / Argon** *(clear the* **Default fluid** *check box)*
Default solid	**Metals / Aluminum**
Wall Conditions	**Default wall radiative surface: Pre-Defined / Real Surfaces / Aluminum, commercial sheet** *Default zero roughness*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	**Result resolution level** *of* `4`

🕮 *Only one semi-transparent solid material, the quartz glass, is used in this device. Its absorption properties are specified as dependent on the wavelength with an abrupt change in absorption at 2500 nm. The UV radiation from the tungsten filament is negligible at 2900K. Thus, a two-bands spectrum with the bands edge at 2500 nm allows to simulate the radiation absorption in the glass components of the lamp accurately enough.*

Adjusting the Computational Domain Size

Specify the computational domain size as follows:

X max = 0.15 m	Y max = 0.2 m	Z max = 0.15 m
X min = 0 m	Y min = -0.12 m	Z min = -0.15 m

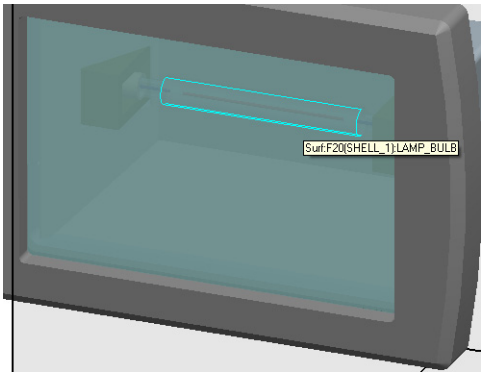
Specify the **Symmetry** condition at **X min**.

Specifying Fluid Subdomain

Halogen lamps are filled with an inert gas and some small amount of halogen (iodine or bromine). For the purposes of this simulation we can consider the lamp as filled with an inert gas only. The gas in a halogen lamp is at the pressure several times higher than atmospheric. We use fluid subdomain to define both the gas filling the lamp and its pressure.

1 Click **Flow Analysis > Insert > Fluid Subdomain**.

2 Select the inner cylindrical surface of the **LAMP.ASM\LAMP_BULB** component. Immediately the fluid subdomain you are going to create is displayed in the graphics area as a body of blue color.



3 Under **Fluids** make sure that **Gases/Real Gases/Steam** is selected in the **Fluid type** list and clear the **Air (Gases)** check box in the list of fluids below, so that only **Argon** remains selected.

4 Under **Thermodynamic parameters** in the **Pressure** box type `2 atm`.

5 Click **OK**.

Specifying Heat and Radiation Conditions

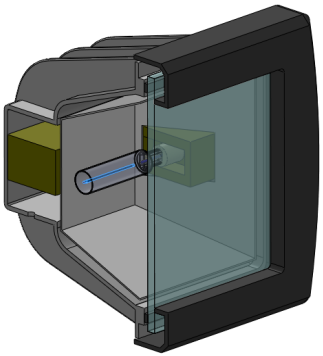
There are several ways to define a heat source in FloEFD. The surface area of the cylindrical straight-line filament can substantially differ from the actual surface area of the coil. If you specify a heat source by its power, this difference must be considered. To avoid discrepancy between the actual and specified radiation heat transfer you can:

a) define a heat source with the temperature specified,
b) then define a radiation source with the power specified.

To do this, we specify a **Volume Heat Source** with the temperature of 2900K. The value of the convective heat transfer rate is determined as a **Surface Goal** and the **Radiation Source** power is defined as 150 Watt minus the convective heat transfer rate. And finally, the outgoing radiation by the filament must be excluded from the calculation, so the filament surface must be defined as a whitebody surface.

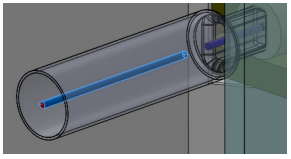
Specify the volume heat source as shown in the table below:

Type	**Volume Heat Source**
Name	**2900 K**
Components to apply	**LAMP.ASM\LAMP_WIRE**
Parameter: **Temperature** of `2900 K`	



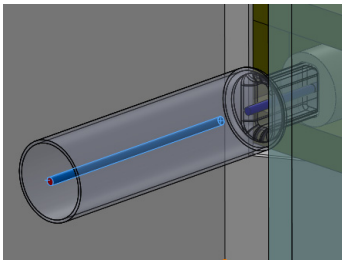
📖 *The true temperature of the filament can be estimated from its color temperature. The typical values of the filament color temperature are specified by the lamp manufacturer. For the filament temperature of about 3000 K the color temperature of tungsten is 2-3 % higher than its equivalent true temperature.*

Specify the goal necessary for calculating the convective heat transfer rate:

GOAL TYPE	GOAL PARAMETER	FACE
Surface Goal	Heat Transfer Rate	The faces of the **LAMP.ASM\LAMP_WIRE** component located within the computational domain. Select the **LAMP.ASM\LAMP_WIRE** component in the Model tree. 

Specifying Radiation Source

Follow the steps below to specify the radiation source:



1 Click **Flow Analysis > Insert > Radiation Source**.

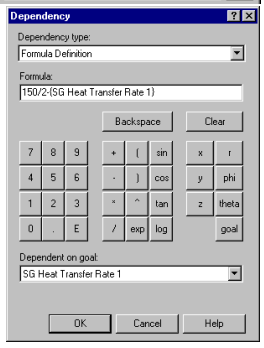
2 In the Model tree select the **LAMP.ASM\LAMP_WIRE** component. All the filament faces are selected as **Faces to Apply the Radiation Source** .

3 Select **Diffusive** as the **Type**.

4 Under **Power** select **Power** and then click **Dependency** .

5 In the **Dependency Type** list select **Formula Definition**. In the **Formula** box, type the formula for the total heat power emitted by the source. To add a goal to the formula, select the goal in the **Dependent on goal** list and click **goal** in the input panel. The resulting expression must be the following:

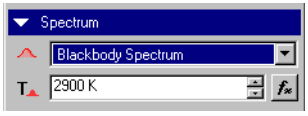
```
150/2-{SG Heat Transfer Rate 1}
```



📖 *We used a **Volume Heat Source** to define the heat transferred from the filament by the convection. To specify the remaining heat power, transferred by the radiation, the calculated heat transfer rate of the volume source must be subtracted from the total heat power.*

6 Click **OK** to return to the **Radiation Source** dialog.

7 Under **Spectrum** select **Blackbody Spectrum** and enter `2900 K` in the **Blackbody Temperature** box.

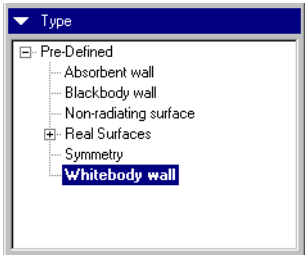


8 Click **OK** .

The new **Diffusive Radiation source 1** item appears in the **Analysis tree**.

Specifying Radiative Surfaces

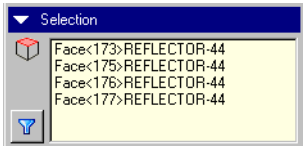
Follow the steps below to specify the radiative surfaces:

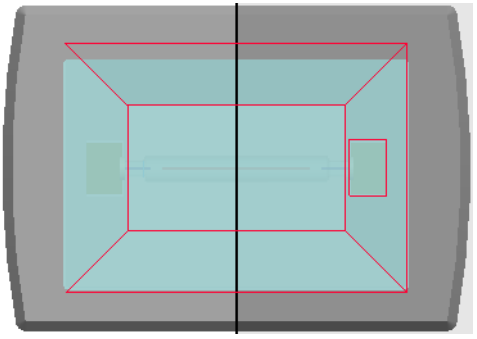


1. Click **Flow Analysis > Insert > Radiative Surface**.
2. Under **Type**, expand the list of **Pre-Defined** radiative surfaces and select **Whitebody wall**.
3. In the FloEFD Analysis Tree select the **Diffusive Radiation source 1** item.
4. Click **OK** . Rename the new **Radiative Surface 1** item to `Radiative Surface Filament`.
   Click anywhere in the graphic area to clear the selection.
5. Click **Flow Analysis > Insert > Radiative Surface**.
6. Under **Type**, click **Create/Edit**.
7. In the **Engineering Database**, under **Radiative Surfaces > User Defined**, create a new item and change its **Name** to `Tutorial Aluminum, polished`.
8. Change the parameters of the surface as shown below:

Radiative surface type	Wall
Specularity coefficient	0.8
Emissivity	Specific for thermal and solar radiation
Emissivity coefficient	0.1
Solar absorptance	0.1

9. Save the created radiative surface and exit the **Engineering Database**.
10. Under **Type**, expand the list of **User-Defined** radiative surfaces and select `Tutorial Aluminum, polished`.
11. Select the inner faces of **REFLECTOR** located (at least, partially) within the computational domain.





12. Click **OK** . Change the name of the new radiative surface to `Radiative Surface Reflector`.

Specifying Solid Materials

For the opaque components, specify the **Solid Material** as following:

LAMP.ASM\LAMP_WIRE	**Pre-Defined\Metals\Tungsten**
LAMP.ASM\LAMP_PINCH (the first one in the tree)	**Pre-Defined\Metals\Molybdenum**
LAMP.ASM\LAMP_BASE (the first one in the tree) **HOLDER**	**Pre-Defined\Ceramics\Alumina (96%)**
SEAL	**Pre-Defined\Glasses and Minerals\Glass Lid Seal**

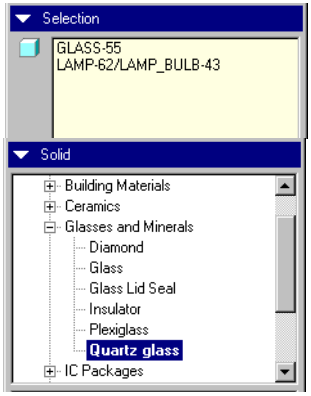
Specifying Bodies and Materials Transparency

Assign the **Quartz glass** material to the bulb and the glass cover and specify these components as semi-transparent to radiation.

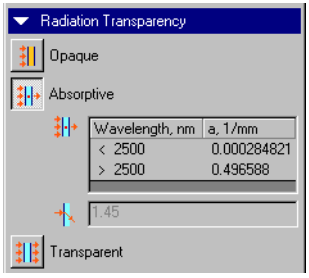
1 Click **Flow Analysis > Insert > Solid Material**

2 In the Model Tree select the **GLASS** and **LAMP.ASM\LAMP_BULB** components.

3 Under **Solid** expand the list of **Pre-Defined** solid materials and select **Quartz Glass** under **Glasses and Minerals**.



4 Under **Radiation Transparency** select **Absorptive**.

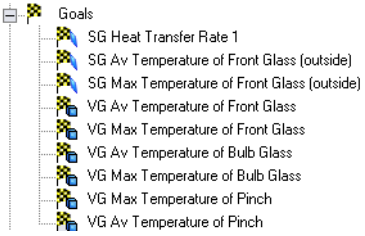


📖 *Absorptive body is semi-transparent. It means that it absorbs the heat radiation within its volume. This option is available only if the absorption coefficient is specified in the solid material definition in the Engineering Database and the* ***Absorption in solids*** *check box is selected under* ***Radiation*** *in the* ***Wizard*** *or* ***General Settings****. The* ***Absorption Coefficient*** *and* ***Refractive Index*** *values are specified in the Engineering Database and are provided here just for reference.*

5 Click **OK**. FloEFD now treats this solid material and all solid bodies it is assigned to as semi-transparent to the thermal radiation.

Specifying Goals

Specify surface goals of the maximum and average temperatures at the outer surface of the **GLASS** component.In addition, specify volume goals of the **GLASS**, **LAMP.ASM\LAMP_BULB** and **LAMP.ASM\LAMP_PINCH**(the first one in the tree) maximum and average temperatures. (you must select **Temperature (Solid**) as the goal parameter). You can rename the goals as shown to make it easier to monitor them during the calculation.


Goals
SG Heat Transfer Rate 1
SG Av Temperature of Front Glass (outside)
SG Max Temperature of Front Glass (outside)
VG Av Temperature of Front Glass
VG Max Temperature of Front Glass
VG Av Temperature of Bulb Glass
VG Max Temperature of Bulb Glass
VG Max Temperature of Pinch
VG Av Temperature of Pinch

Setting Local Initial Mesh

It makes sense to adjust the computational mesh to better resolve the semi-transparent solid bodies and the fine filament. The most convenient way to do this is to specify **Local Initial Mesh** - it allows us to obtain more accurate solution in these specific regions without creating an excessively fine mesh in other regions.

1. In the Model Tree select the filament and bulb components of the halogen lamp (**LAMP.ASM\LAMP_WIRE, LAMP.ASM\LAMP_BULB**).
2. Click **Flow Analysis > Insert > Local Initial Mesh**.

*In Creo Parametric click **Flow Analysis > Mesh > Local Initial Mesh**.*

3. Clear the **Automatic settings** check box.
4. Go to the **Solid/Fluid Interface** tab and set **Small solid features refinement level** to 1.
5. Go to the **Refining Cells** tab. Select **Refine solid cells** and set **Level of refining solid cells** to 3.
6. Go to the **Narrow Channels** tab. Select **Enable narrow channels refinement** and set **Characteristic number of cells across a narrow channel** to 7.
7. Click **OK** to save local initial mesh settings.

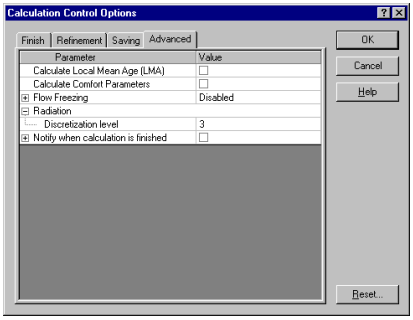
Adjusting the Calculation Control Options

1 Click **Flow Analysis > Calculation Control Options**.

*In Creo Parametric click **Flow Analysis > Project > Calculation Control Options**.*

2 Switch to the **Advanced** tab.

3 Under **Radiation**, make sure that the value of **Discretization level** is set to `3`. This value is appropriate for the given conditions and allows to obtain an acceptable accuracy in the case of compact radiation sources.



*The **Discretization level** controls the discretization of the whole directional domain into equal solid angles or directions. The higher the discretization level, the better the accuracy, but the more CPU time and memory resources are required for the calculation.*

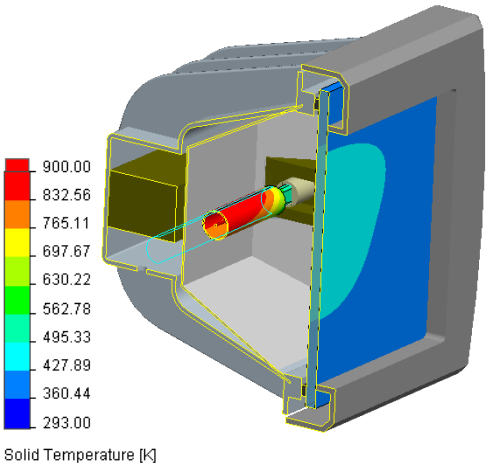
4 Click **OK**.

Save the model and run the calculation.

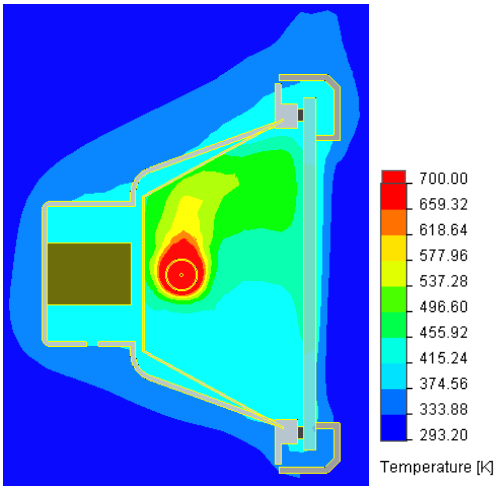
Results

In accordance with the obtained results, we can say that the glass cover and the lamp bulb operate at permissible temperatures.

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value
SG Av Temperature of Front Glass (outside)	[K]	406,9207831	405,9210302	404,5090822	406,9207831
SG Max Temperature of Front Glass (outside)	[K]	447,7389048	446,1168752	443,7679985	447,7389048
VG Av Temperature of Front Glass	[K]	408,4118527	407,4064091	405,9832431	408,4118527
VG Max Temperature of Front Glass	[K]	450,5026033	448,8672748	446,5126887	450,5026033
VG Av Temperature of Bulb Glass	[K]	711,9991158	711,2355085	710,1377058	711,9991158
VG Max Temperature of Bulb Glass	[K]	909,362173	908,7665526	907,8839731	909,362173
VG Av Temperature of Pinch	[K]	496,2168727	495,7868545	494,6775118	496,6216646
VG Max Temperature of Pinch	[K]	520,884922	520,3716383	519,4545268	521,1030745



The glass temperature distribution (surface plot of solid temperature) in the range from 293 to 900 K.



The temperature distribution in the symmetry plane (cut plot of temperature) in the range from 293 to 700 K.

D2

Hospital Room

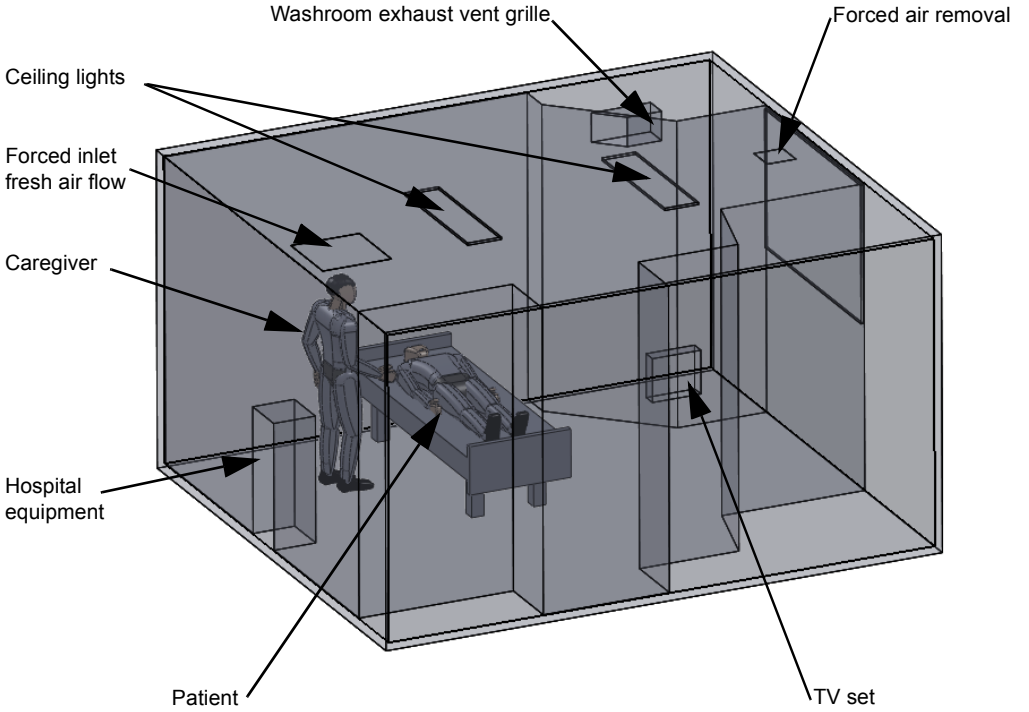
Some of the features used in this tutorial are available for the HVAC module users only.

Problem Statement

This tutorial demonstrates the capability of FloEFD to predict the performance of a building ventilation system and to estimate air quality and general thermal sensation by calculating comfort criteria. It is shown how to define a project, i.e. specify the heat sources, boundary conditions and calculation goals, and how to obtain values of comfort criteria.

Here we consider a hospital isolation room and estimate the ventilation system effectiveness with respect to the contaminant removal and thermal satisfaction of people in the room. A typical patient room includes standard features such as a patient bed, exhausts, lightening, equipment. The overhead ventilation system contains an overhead ceiling supply diffuser, the ceiling and the washroom exhausts. The contaminant source is assumed to be the patient breathing. The heat sources are lights, a medical equipment, a TV, a patient and a caregiver.

The ventilation system and the patient room features are shown at the figure below.



The following parameters are used to estimate the ventilation system effectiveness with respect to contaminant removal: Contaminant Removal Effectiveness (CRE) and Local Air Quality Index (LAQI).

The following parameters are used to estimate the ventilation system effectiveness with respect to thermal satisfaction of people: Air Diffusion Performance Index (ADPI), Predicted Mean Vote (PMV) and Predicted Percent Dissatisfied (PPD).

Model Configuration

Copy the **D2 - Hospital Room** folder into your working directory. Open the **hospital_room.asm** assembly.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **hospital_room.asm** assembly located in the **D2 - Hospital Room\Ready To Run** folder and run the desired projects.*

Project Definition

Using the **Wizard** create a new project as follows:

Project name	*Hospital room*
Configuration	*Use current*
Unit system	**SI**, *units for* **Temperature**: **°C (Celsius)**
Analysis type	**Internal**, **Exclude cavities without flow conditions**
Physical features	**Gravity***:* **Y component** *of* `-9.81 m/s^2`
Default fluid	**Gases / Air** **Gases / Expired Air** *(user-defined)* *Click* **New** *and in the Engineering Database create a new item named* `Expired Air` *by copy-pasting the pre-defined* **Air,** *available under* **Materials\Gases\Pre-Defined***, to the* **Materials\Gases\User Defined** *folder*
Wall Conditions	*Default*
Initial Conditions	**Thermodynamic parameters***:* **Temperature** *of* `19.5°C` **Concentration***:* **Mass fraction** *of* **Air** *is* `1` **Mass fraction** *of* **Expired air** *is* `0`
Result and Geometry Resolution	**Default Result resolution level** *of 3*

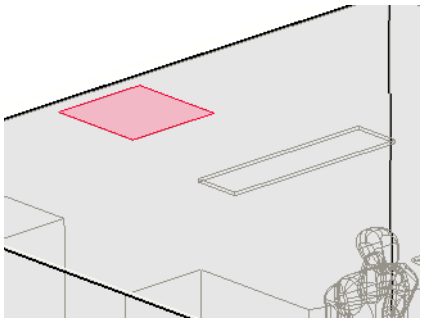
After creating the project an error message appears informing you that the fluid volume recognition has failed. Select **No** to ignore the closing openings with **Create Lids** tool. The problem disappears after disabling the **FLUIDVOLUMEPART** component in the **Flow Analysis > Component Control** dialog box to treat it as a fluid region.

Rebuild the project by clicking **Flow Analysis >Project > Rebuild**.

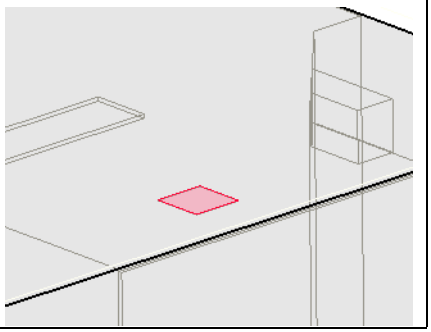
Boundary Conditions

Specify the inlet and outlet boundary conditions as shown in the tables below:

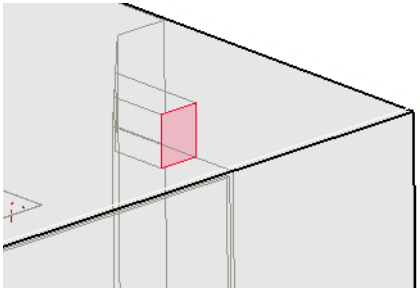
Type	**Inlet Volume Flow**
Name	**Inlet Volume Flow 1**
Faces to apply	the inner face of the **ROOM** component (the one above the hospital equipment table)
Forced inlet fresh air flow Parameters: **Volume Flow Rate** of `4.8 m^3/min`	

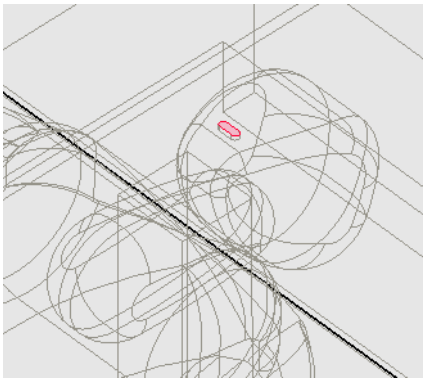


Type	**Outlet Volume Flow**
Name	**Outlet Volume Flow 1**
Faces to apply	the inner face of the **ROOM** component (the one near the window)
Forced air removal Parameters: **Volume Flow Rate** of `2.6 m^3/min`	



Type	**Environment Pressure**
Name	**Environment Pressure 1**
Faces to apply	the inner face of the **ROOM** component, as shown
Washroom exhaust vent grille **Thermodynamic Parameters**: Default values (101325 Pa and 19.5°C)	



Type	**Inlet Volume Flow**	
Name	**Inlet Volume Flow 2**	
Faces to apply	a face of the **PATIENT** component, representing the patient's mouth, as shown	
Contaminated expired air Parameters: **Volume Flow Rate** of `12 l/min` **Substance Concentrations**: **Mass fraction** of **Air** is 0 **Mass fraction** of **Expired Air** is 1		

Specifying Heat Sources

There are several heat sources in the hospital room: ceiling lights, a TV set and hospital equipment. The caregiver and the patient are the sources of heat also. The amount of heat produced by a human body depends on the kind of activity the person is involved in. A patient laying on the bed produces significantly less heat than a caregiver, whose work requires physical activity and concentration.

Since we do not consider heat conduction in solids in this simulation, we use surface heat sources with the fixed heat transfer rate.

Click **Flow Analysis > Insert > Surface Source**.

1 In the Model tree, select the **PATIENT.ASM** component. This component appears in the **Faces to Apply the Surface Source** list.

2 Under **Parameter** specify **Heat Transfer Rate** of `81 W`.

3 Click **OK**.

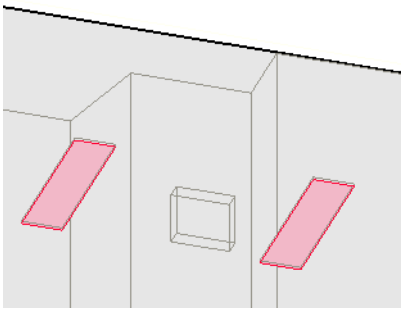
Rename the created heat source to Patient.

In the same way specify the surface heat source of `144 W` at all faces of the **CAREGIVER.ASM** component.

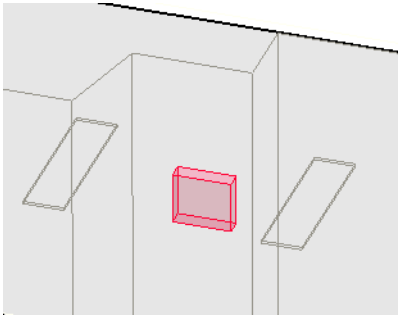
Rename the created heat source to Caregiver.

Other sources of heat are not represented by separate components, but by cuts and extrudes made on the **Room** component. Use the tables below as a reference to specify the remaining heat sources:

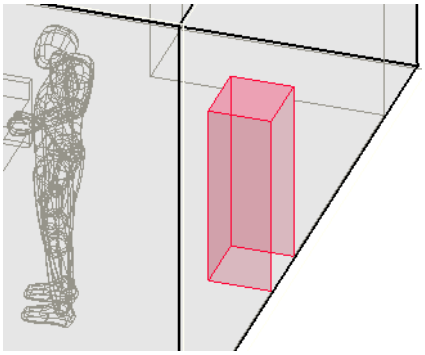
Type	**Surface Heat Source**
Name	**Ceiling Lights**
Faces to apply	both inner faces of the **ROOM** component representing the ceiling lights
Parameters: **Heat Transfer Rate** of `120 W`	



Type	**Surface Heat Source**
Name	**TV Set**
Faces to apply	all inner faces of the **ROOM** component representing the TV set
Parameters: **Heat Transfer Rate** of `50 W`	



Type	**Surface Heat Source**
Name	**Hospital Equipment**
Faces to apply	all inner faces of the **ROOM** component representing the hospital equipment
Parameters: **Heat Transfer Rate** of `50 W`	



Specifying Calculation Control Options

By default, calculation of comfort parameters is disabled in FloEFD to save the CPU time and memory resources. Besides comfort parameters, FloEFD is capable of calculating Local Mean Age (LMA) and Local Air Change Index (LACI) parameters:

- LMA is the average time for fluid to travel from the selected inlet opening to the point considering both the velocity and diffusion.
- LACI (Local Air Change Index) is the ratio of the V/Q value, where V is the computational domain fluid volume and Q is the volume flow rate of the fluid entering this volume, to the average time τ for the fluid to travel from the selected inlet opening to the point considering both the velocity and diffusion.

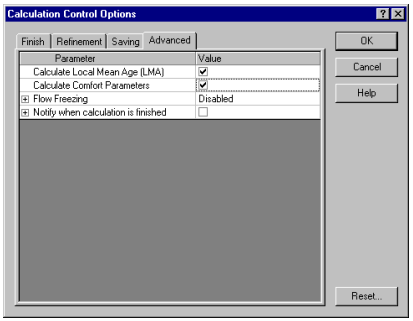
Calculation of comfort parameters, LMA and LACI can be enabled in the **Calculation Control Options** dialog.

1 Click **Flow Analysis > Calculation Control Options**.

In Creo Parametric click **Flow Analysis > Project > Calculation Control Options**.

2 Switch to the **Advanced** tab.

3 Select the **Calculate Local Mean Age (LMA)** and **Calculate Comfort Parameters** check boxes.

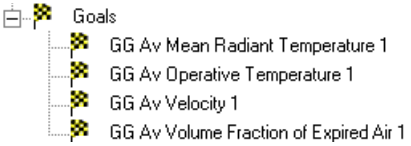


Selecting **Calculate Local Mean Age (LMA)** *check box enables calculation of LMA, Dimensionless LMA and LACI.*

4 Click **OK**.

Specifying Goals

Specify global goals of **Av Mean Radiant Temperature**, **Av Operative Temperature**, **Av Velocity** and **Av Volume Fraction of Expired Air**.


Goals
GG Av Mean Radiant Temperature 1
GG Av Operative Temperature 1
GG Av Velocity 1
GG Av Volume Fraction of Expired Air 1

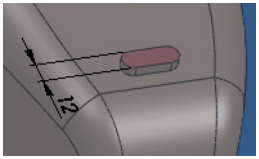
You can use **Mean Radiant Temperature** *and* **Operative Temperature** *as the goal parameters only after you enable calculation of comfort parameters in the* **Calculation Control Options** *dialog.*

Adjusting Initial Mesh

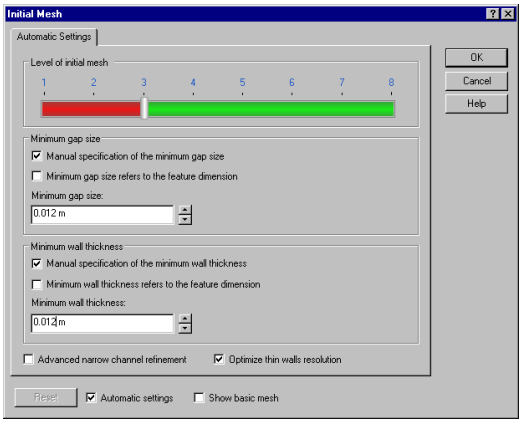
1 Click **Flow Analysis > Initial Mesh**.

In Creo Parametric click **Flow Analysis > Mesh > Initial Mesh**.

2 Select the **Manual specification of the minimum gap size** check box.
You can see that FloEFD determined the **Minimum gap size** value as 0.012 m, which is equal to the width of the face representing the patient's mouth (12 mm).



3 Select the **Manual specification of the minimum gap wall thickness** check box.
You can see that the value of **Minimum wall thickness**, determined by FloEFD, is too small, which can lead to unnecessary mesh refinement and increased amount of memory required for calculation.
To avoid this, specify the value of **Minimum wall thickness** equal to **Minimum gap size** of 0.012 m.



4 Click **OK** to save the initial mesh settings and close the dialog.

Setting Local Initial Mesh

To better resolve the complex geometry of the **CAREGIVER** and **PATIENT** components and more accurately account the heat produced by the heat sources specified at these components, we employ the local initial mesh refinement.

1 Click **Flow Analysis > Insert > Local Initial Mesh**.

In Creo Parametric click **Flow Analysis > Mesh > Local Initial Mesh**.

2 In the Model Tree, select the **CAREGIVER.ASM** and **PATIENT.ASM** components.

3 Clear the **Automatic settings** check box. Go to the **Solid/Fluid Interface** tab.

4 Set **Small solid features refinement level** to 4.

5 Go to the **Refining Cells** tab. Select **Refine fluid cells** and set **Level of refining fluid cells** to 2.

6 Click **OK** to save local mesh settings.

Run the calculation. During the calculation process you may notice that the **Av Volume Fraction of Expired Air** goal converges slower than the other goals specified. Since this is a tutorial example, there may be not need to wait before the solution fully converges. To save the CPU time, you can stop the calculation earlier, for example when all the other specified goals converge.

Results

Overview of Comfort Parameters

It is a common practice to assess the performance of a ventilation system by some standard criteria, named comfort parameters. With FloEFD you can simulate various environments and get the values of comfort parameters, determining whether the air quality and temperature are safe and comfortable for people working or living in these environments. Later we will use FloEFD results processing tools to see and analyze the values of comfort parameters obtained in the calculation.

The following two parameters are used to assess the ventilation system effectiveness in contaminated air removing:

- **Contaminant Removal Effectiveness (CRE)**. This parameter is an index that provides information on the effectiveness of a ventilation system in removing contaminated air from the whole space. For a perfect mixing system CRE = 1. Values above 1 are good, values below 1 are poor.
- **Local Air Quality Index (LAQI)** is an index that provides information on the effectiveness of a ventilation system in removing contaminated air from a local point.

The following several parameters are used to estimate the ventilation system effectiveness with respect to the thermal satisfaction of people in the ventilated area:

- **Mean Radiant Temperature (MRT)** is the uniform surface temperature of an imaginary black enclosure in which an occupant would exchange the same amount of radiant heat as in the actual non-uniform space.
- **Operative Temperature** is the uniform temperature of an imaginary black enclosure, in which an occupant would exchange the same amount of heat by radiation plus convection as in the actual non-uniform environment.
- **Draft Temperature** is the difference in temperature between any point in the occupied zone and the control condition. *"Draft"* is defined as any localized feeling of coolness or warmth of any portion of the body due to both air movement and air temperature, with humidity and radiation considered constant.
- **Air Diffusion Performance Index (ADPI)** is the percentage of the space in which the air speed is less than 0.35 m/s and the Draft Temperature falls between -1.7 °C and 1.1 °C.

  📖*Note:If Draft Temperature or ADPI is calculated as Volume Parameters, the reference space or zone is the specified volume region. In all other cases the whole computational domain is considered.*

- **Predicted Mean Vote (PMV)** is an index that predicts the mean value of the votes of a large group of persons on the 7-point thermal sensation scale, based on the heat

balance of the human body. Thermal balance is obtained when the internal heat production in the body is equal to the loss of heat to the environment.

cold	cool	slightly cool	neutral	slightly warm	warm	hot
-3	-2	-1	0	+1	+2	+3

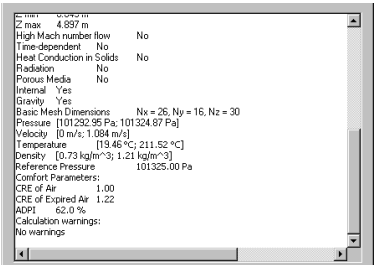
- **Predicted Percent Dissatisfied (PPD)** is an index that provides information on thermal discomfort or thermal dissatisfaction by predicting the percentage of people likely to feel too warm or too cool in a given environment.

Obtaining CRE Value

You can see the calculated value of the Contaminant Removal Effectiveness (CRE) in the calculation results summary.

In the Analysis tree, right-click the **Results** icon and select **Summary**.

You can see the **CRE of Expired Air** value at the bottom of the **Results Summary** page, in the **Comfort Parameters** section. The value of **CRE of Expired Air** is higher than 1, which means that the ventilation system is reasonably effective in removing the contaminated air.


Z max 4.897 m
High Mach number flow No
Time-dependent No
Heat Conduction in Solids No
Radiation No
Porous Media No
Internal Yes
Gravity Yes
Basic Mesh Dimensions Nx = 26, Ny = 16, Nz = 30
Pressure [101292.95 Pa; 101324.87 Pa]
Velocity [0 m/s; 1.084 m/s]
Temperature [19.46 °C; 211.52 °C]
Density [0.73 kg/m^3; 1.21 kg/m^3]
Reference Pressure 101325.00 Pa
Comfort Parameters:
CRE of Air 1.00
CRE of Expired Air 1.22
ADPI 62.0 %
Calculation warnings:
No warnings

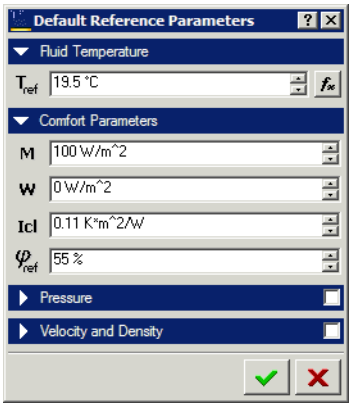
Volume Parameters

We can obtain the values of thermal satisfaction parameters with the **Volume Parameters** results processing feature. The volume, in which the parameters will be calculated, is **FLUIDVOLUMEPART** component (i.e. the entire fluid region within the computational domain).

Before specifying **Volume Parameters**, we need to check the values of reference parameters: metabolic rate, external work, closing thermal resistance and relative humidity, used to calculate comfort parameters such as PMV and PPD. These reference parameters define the approximate heat power produced by a human body depending on the activity and health condition, insulating properties of the closing and humidity of the air.

1 Click **Flow Analysis > Results > Default Reference Parameters**.

2 Specify **Metabolic rate** M of `100 W/m^2`. Keep the other values default.



🕮 *The closing thermal resistance of 0.11 K·m^2/W corresponds to a light working ensemble: light underwear, cotton work shirt with long sleeves, work trousers, woolen socks and shoes. The definition of clothing insulation relates to heat transfer from the whole body and, thus, also includes the uncovered parts of the body, such as head and hands.*

🕮 *The relative humidity of 55% is typical for indoor conditions. If the relative humidity is considered in the analysis (the* **Humidity** *option is selected in the* **General Settings***), the actual calculated value of the relative humidity is used as the reference parameter.*

3 Click **OK**.

Now we can use the **Volume Parameters** feature to see the values of comfort parameters.

1 In the FloEFD Analysis tree right-click the **Volume Parameters** icon and select **Insert**.

2 In the Model tree, select the **FLUIDVOLUMEPART** component.

3 Under **Parameters** click **More Parameters**. The **Customize Parameter List** dialog appears.

4 Expand the **Comfort Parameters** item and select the following parameters:

- **Mean Radiant Temperature**,
- **Operative Temperature**,
- **PMV**,
- **PPD**,
- **Draft Temperature**
- **LAQI of Air**,
- **LAQI of Expired Air**.

5 Click **OK** to close the **Customize Parameter List** dialog.

6 In the **Volume Parameters** dialog make sure that the selected parameters are also selected as the **Parameters to Evaluate** under **Parameters**. Additionally select the **ADPI** parameter.

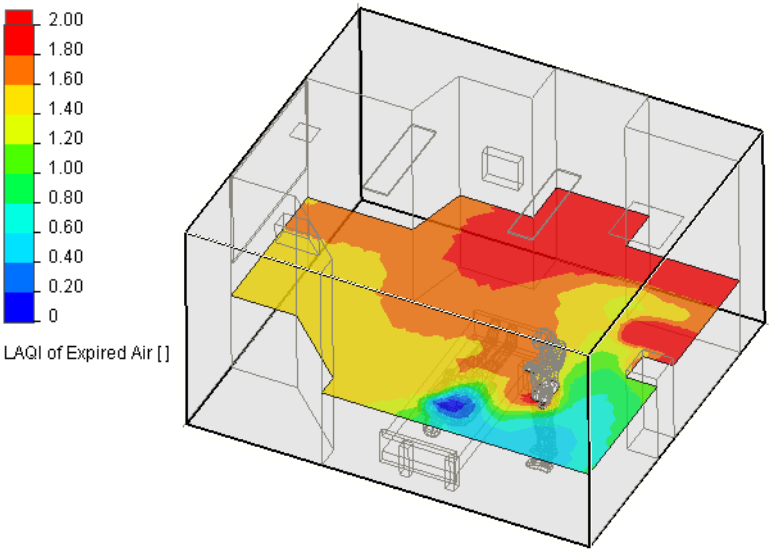
7 Click **Export to Excel**. A spreadsheet with the selected parameters values appears.

Parameter	Average	Bulk Average	Volume [m^3]
Mean Radiant Temperature [°C]	24,0880572	24,0841671	41,858273
Operative Temperature [°C]	23,5721111	23,5684728	41,858273
PMV [ ]	0,731470357	0,730771361	41,858273
PPD [%]	17,4027619	17,3802028	41,858273
Draft Temperature [K]	0,726633855	0,72332388	41,858273

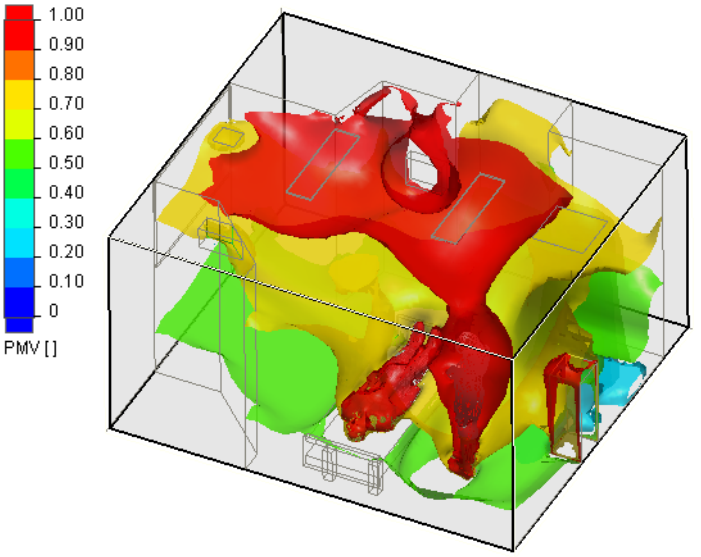
Parameter	Value
ADPI [%]	62,0554295

Cut Plots and Isosurfaces

To see how the quality of air with respect to the contained contaminant changes through the room, we create a cut plot by the **LAQI of Expired Air** parameter at the distance of 1 m from the floor - i.e. slightly above the level of the patient's head. The higher the value, the less the concentration of the contaminant and better it is removed.



The isosurfaces of **PMV** at 0, 0.25, 0.5, 0.75 and 1 allows us to estimate the level of thermal comfort through the room - from 0 (normal) to +1 (slightly warm).



D3

Pollutant Dispersion in the Street Canyon

Some of the features used in this tutorial are available for the HVAC module users only.

Problem Statement

This tutorial demonstrates the capability of FloEFD to simulate an urban pollution. It is shown how to obtain a flow field, and how to define a tracer source to simulate the pollutant dispersion.

Here we consider the street canyon, a road and its flanking buildings shown in **Ref.1**. The street is 50 m wide and is flanked on the North side with 100 m high tower block (31 stories) and on the South side with 30 m high building (10-11 stories). Orientation of the street is east-west.

The source of pollution comes from the vehicular exhaust. Traffic-related air pollution is a complex mixture of gases and particles. Here we consider Nitrogen Oxide (NOx) air pollution. Nitrogen Oxides include NO and NO_2, the former being produced in far more abundance by vehicles than the latter. However, NO rapidly oxidizes to NO_2 which destroys lung tissue with the obvious negative implications for asthmatics. The emission of NOx is assumed to be $8.6 \cdot 10^{-5}$ kg/s.

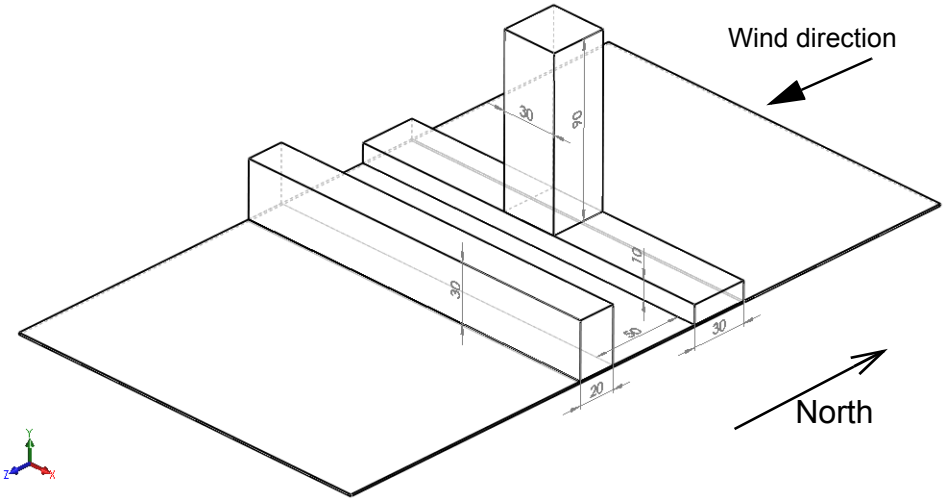
Traffic pollution in a street canyon is characterized by large vertical variability, which is not only related to the variation in the traffic amount but is also influenced by the meteorological conditions.

It is common in engineering practice to describe the wind profile with a power-law in non-complex terrain up to a height of about 200 m above ground level:

$$V(h) = V_r(h/h_r)^p$$

where *V(h)* is the wind speed at height *h*, and V_r is the known wind speed at a reference height h_r. The power-law exponent *p* is an empirically derived coefficient that varies dependent upon the stability of the atmosphere from about 0.1 to about 0.6. The larger the power-law exponent the larger the vertical gradient in the wind speed.

The street canyon is shown at the figure below.



Model Configuration

Copy the **D3 - Street Canyon** folder into your working directory. Open the **tower_building.asm** assembly.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **tower_building.asm** assembly located in the **D3 - Street Canyon\Ready To Run** folder and run the project.*

Project Definition

Using the **Wizard** create a new project as follows:

Project name	*Street Canyon*
Configuration	*Use current*
Unit system	**SI**
Analysis type	**External**
Physical features	**Time-dependent**
Default fluid	**Gases / Air**
Wall Conditions	*Default conditions*
Initial Conditions	**Velocity Parameters**: **Velocity in Z direction** = `2.6*(y/10)^0.25 m/s` **Turbulence Parameters**: **Parameters: Turbulence energy and dissipation** **Turbulence energy** = `0.596 J/kg` **Turbulence dissipation**= `0.423^3/(0.39*y) W/kg` *Other conditions are default*
Result and Geometry Resolution	*Result resolution level of 5* *Other conditions are default*

Adjusting the Computational Domain Size

Specify the computational domain size as follows:

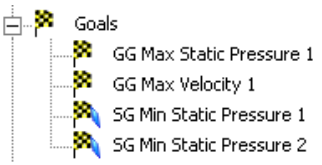
X max = 65 m	Y max = 150 m	Z max = 195 m
X min = 0 m	Y min = -0.005 m	Z min = -164 m

Specify the **Symmetry** condition at **X max** and **X min**.

Specifying Goals

Specify global goals of **Max Static Pressure** and **Max Velocity**.

In addition, specify separate surface goals of **Min Static Pressure** at the leeward walls of the buildings.



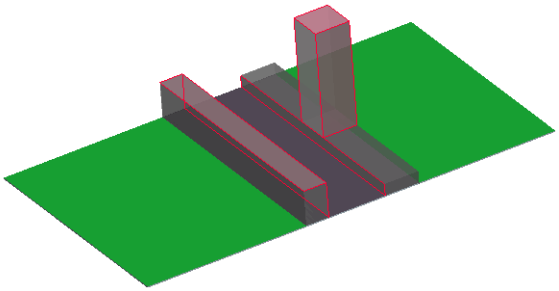
Setting Local Initial Mesh

To better resolve the street canyon and the area around the tower block, we employ the local initial mesh refinement.

1 Click **Flow Analysis > Insert > Local Initial Mesh**.

*In Creo Parametric click **Flow Analysis > Mesh > Local Initial Mesh**.*

2 In the graphics area, select the roof and facing faces of the both buildings and the side face of the tower.



3 Clear the **Automatic settings** check box.

4 Go to the **Refining Cells** tab. Select **Refine fluid cells** and set **Level of refining fluid cells** to 3.

5 Go to the **Narrow Channels** tab and set **Characteristic number of cells across a narrow channel** to 20.

6 Click **OK** to save local mesh settings.

Adjusting the Calculation Control Options

1 Click **Flow Analysis > Calculation Control Options**.

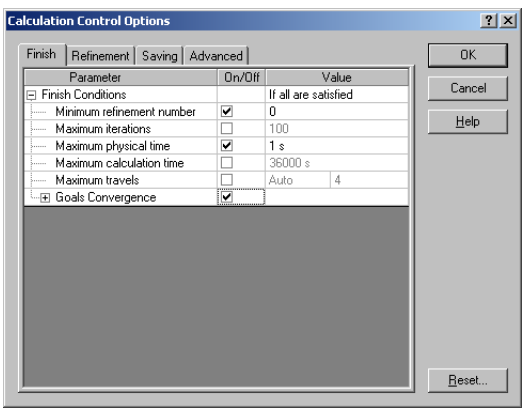
*In Creo Parametric click **Flow Analysis > Project > Calculation Control Options**.*

2 In the **Calculation Control Options** dialog box, change the **Value** of **Finish Conditions** to **If all are satisfied** and select **Goals Convergence**..

3 Click **OK**.

4 Save the model and run the calculation.

Save the model and run the calculation.



Specifying Tracer Study

🕮 *Tracers allow you to study the flow of a certain admixture (tracer) in the existing carrier fluid under assumption that the presence of the admixture has a negligible influence on the carrier fluid flow. Such assumption is justified if the concentration (mass fraction) of the admixture is relatively low.*

Follow the steps below to specify the tracer substance:

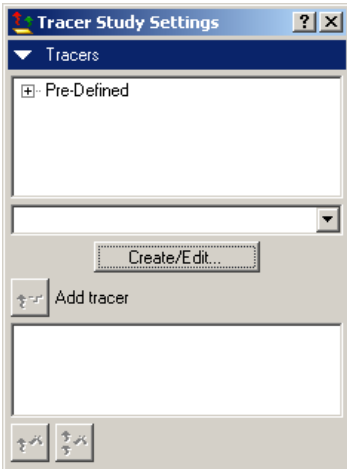
1 Click **Flow Analysis > Insert > Tracer Study**.

2 Under **Tracers**, click **Create/Edit**.

3 In the **Engineering Database**, under **Tracers, User Defined**, create a new item and change its **Name** to `Tutorial NOx`.

4 Change the parameters of the surface as shown below:

Property	Value
Name	Tutorial NOx
Comments	
Molar mass	0.02896 kg/mol
Diffusion defined by	Diffusion coefficient
Diffusion coefficient	26000 cm^2/s
Saturation pressure	☐

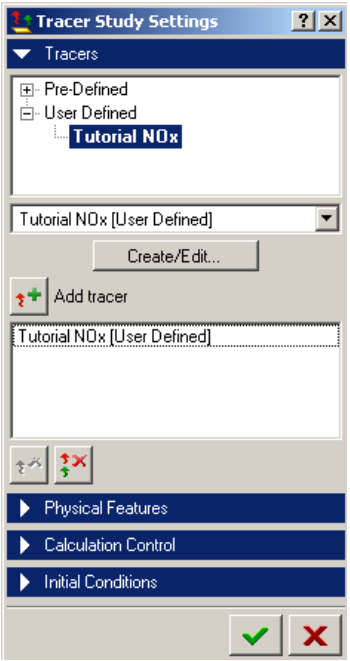


5 Save the created tracer and exit the **Engineering Database**.

6 Under **Tracers**, expand the list of **User-Defined** tracers and select `Tutorial NOx` and click **Add tracer** .

7 The `Tutorial NOx [User Defined]` item appears in the list below.

8 Click **OK** . The new **Tracer Study 1** item appears in the **Analysis tree**.



Follow the steps below to specify the tracer source:

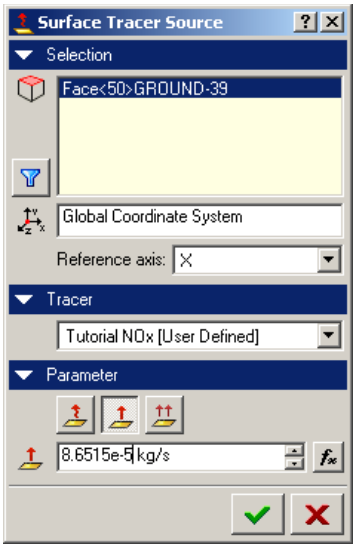
1 Right-click the **Tracer Study 1** icon in the FloEFD Analysis tree and select **Insert Surface Source**. The **Surface Tracer Source** dialog appears.

2 In the graphics area select the road surface. The selected face appears in the **Faces to Apply the Surface Tracer Source** list.

3 Under **Tracer** make sure that `Tutorial NOx` is selected.

4 Under **Parameter** select **Mass Flow** and set its value equal to `8.6515e-5 kg/s`.

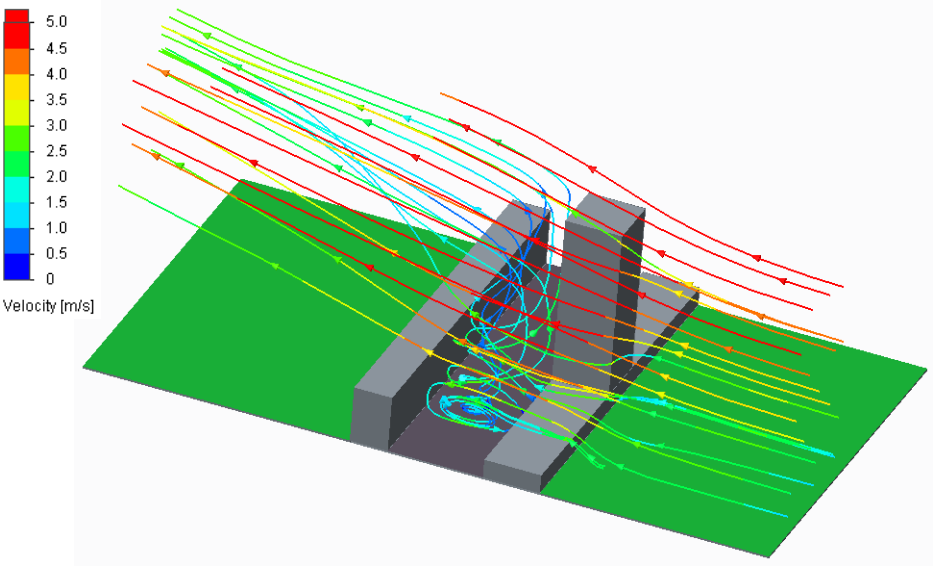
5 Click **OK**. The new **SS Mass Flow of Tutorial NOx 1** item appears in the **Analysis tree**.



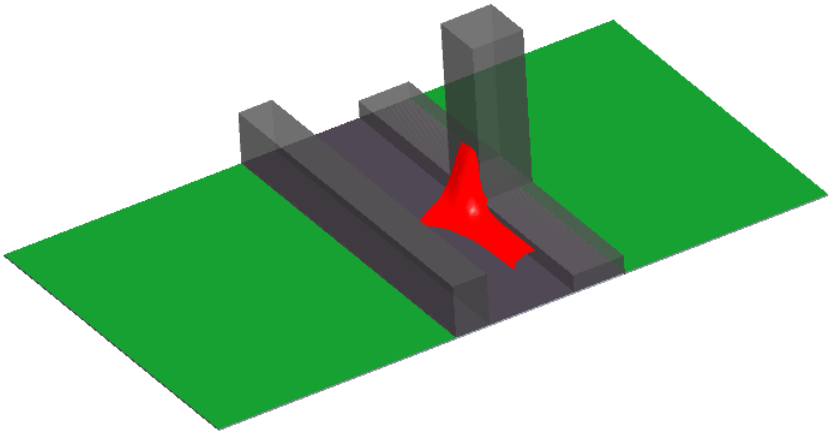
To run the tracer calculation, right-click the **Tracer Study 1** icon in the FloEFD Analysis tree and select **Run**.

Results

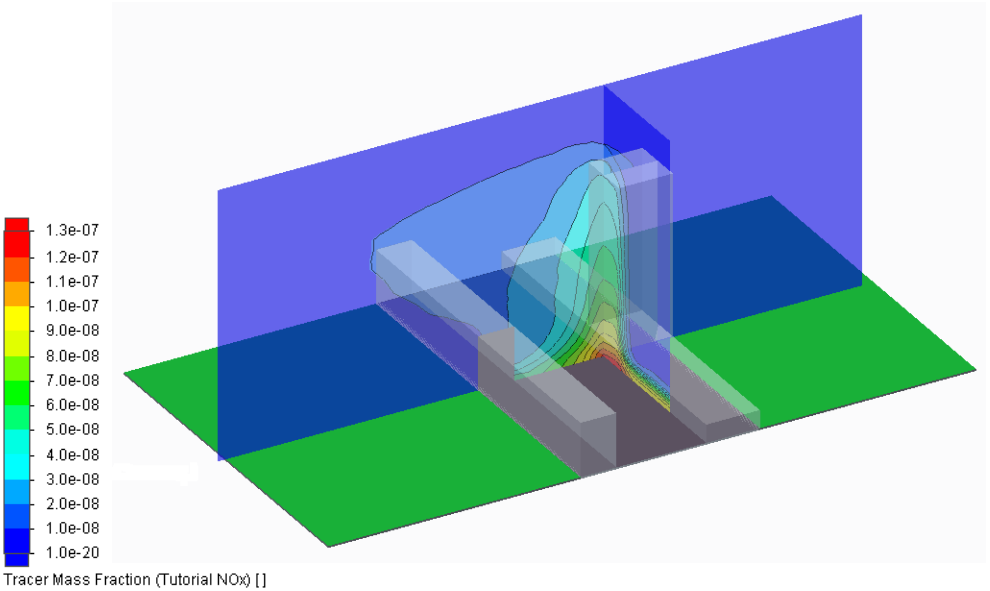
To see how the wind flows around the buildings we will display the **Flow Trajectories**.



The isosurfaces of the **Tracer Mass Fraction (Tutorial NOx)** at $6.4 \cdot 10^{-8}$ allows us to estimate areas of high traffic pollution.



Additionally, let us see the distribution of the **Tracer Mass Fraction (Tutorial NOx)** in the symmetry plane and on the leeward face of the tower block.



1 Qin Y., Kot S.C. *Validation of computer modeling of vehicular exhaust dispersion near a tower block*. Journal of Building and Environment, vol. 25, No2, 1990, pp 125-131.

E

Examples for Electronics Cooling and LED Modules

The examples for **Electronics Cooling** and **LED** modules presented below demonstrate how to use capabilities and features of this module to simulate a wide variety of electronic components. This functionality is available for the Electronics Cooling or LED modules users only.

E1 - Electronic components

E2 - LED lighting

Examples for Electronics Cooling and LED Modules:

E1

Electronic Components

Some of the features used in this tutorial are available for the Electronics Cooling module users only.

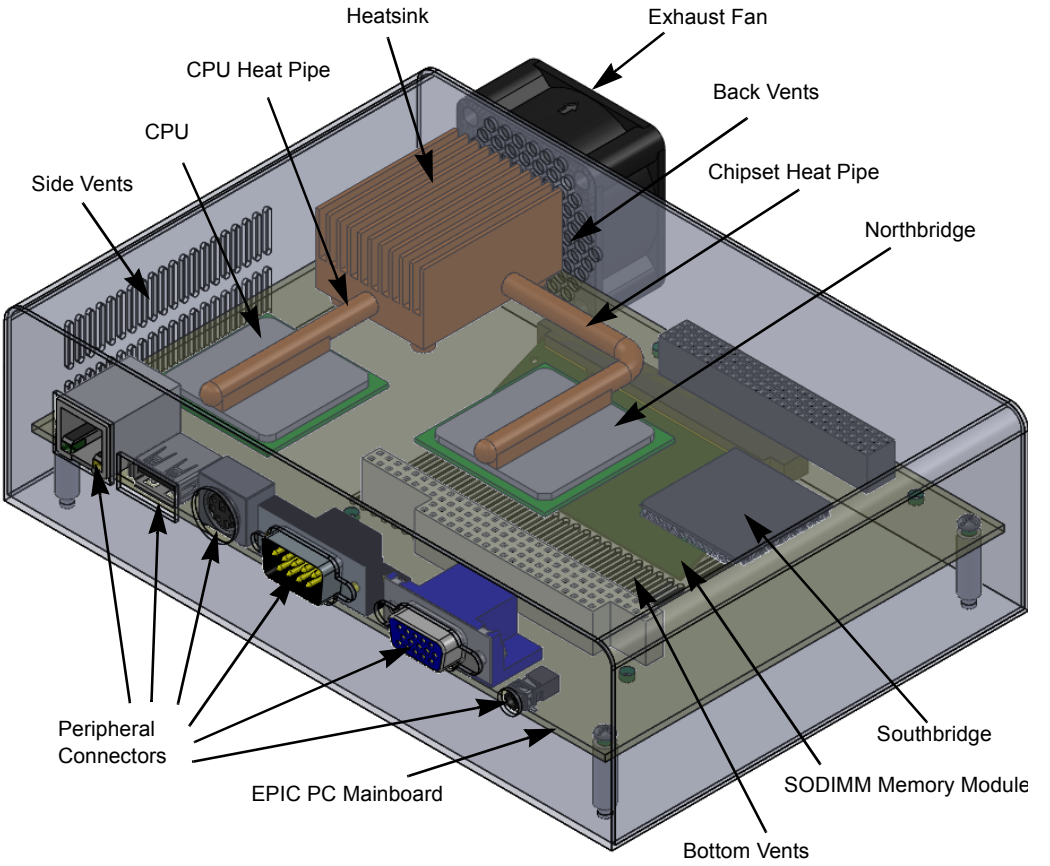
Problem Statement

This tutorial demonstrates the capabilities of FloEFD to simulate cooling of electronic components in an embedded industrial computer by using various features implemented in the **Electronics** module. Here we consider a single board computer with a case, which contains, among other components, CPU, chipset (Northbridge and Southbridge), heat sink with two heat pipes, PCI and ISA slots for a PC104 expansion board, SODIMM slot with memory installed and peripheral connectors.

Air at room temperature enters the case through the vents located at the side and bottom panels and exits through the vents located at the back panel, where an exhaust fan is installed. The resulting flow inside the case removes the heat produced by electronic components (CPU, Northbridge, Southbridge and DDR RAM chips). The heat pipes also transfer the heat produced by CPU and Northbridge to the heat sink, which dissipates it into the air. In the considered model, this heat sink is placed near the exhaust fan.

The objective of the simulation is to ensure that under these conditions, electronic components operate at moderate temperatures. In the table below, you can see the typical values of maximum operating temperatures of the electronic components under consideration.

Electronic component	Maximum operating temperature
CPU	85 °C
Northbridge	80 °C
Southbridge	100 °C
DDR RAM chip	85 °C

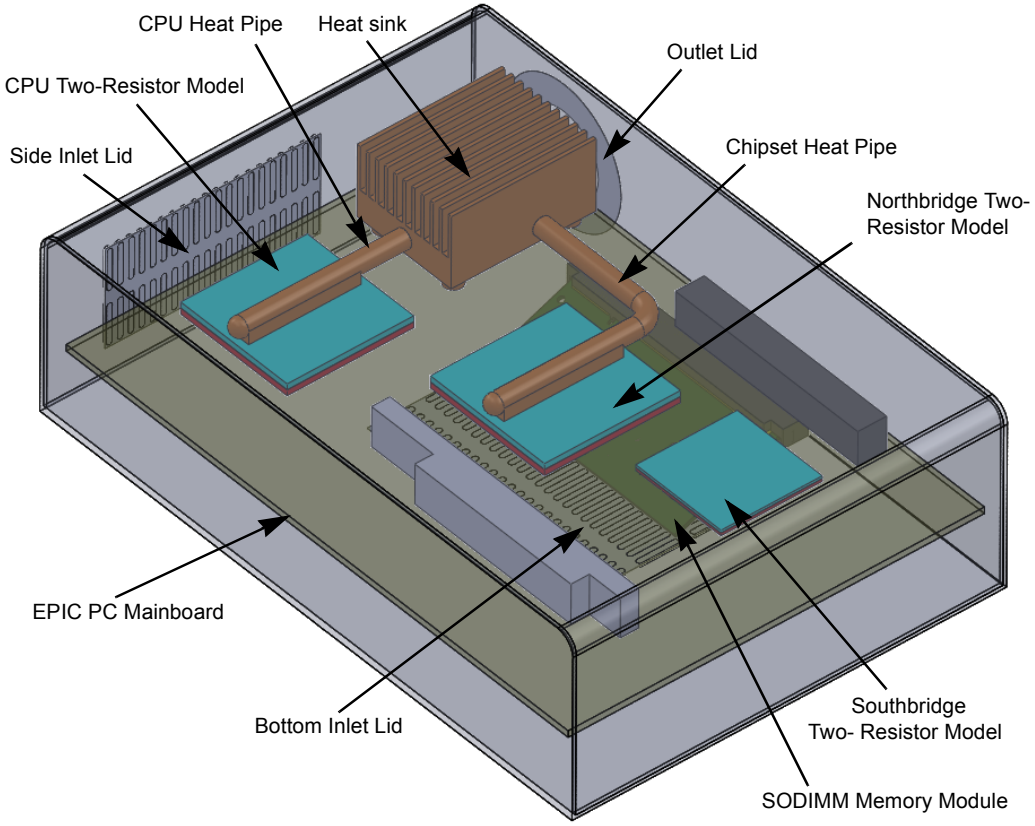


Opening the Model

Copy the **E1 - Electronic Components** folder into your working directory. Open the **epic_pc.asm** assembly. Select the generic instance. This is the original model geometry in accordance with the problem statement. After studying this model, switch to the **SIMULATION_MODEL** instance.

To unhide simplified components components, click **View > Visibility > Unhide All**.

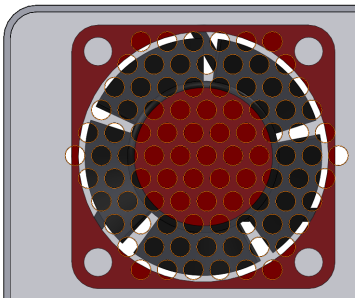
🕮 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the* ***epic_pc.asm*** *assembly located in the* ***E1 - Electronic Components\Ready To Run*** *folder and run the project.*



Simulation model

To simplify the problem for this tutorial and, therefore, to save your computer resources, we neglect some particular components and features, which do not affect the flow and heat exchange much. These include holes in PCI and ISA slots, screws and peripheral connectors. The model geometry of exhaust fan is also excluded from the simulation and is replaced by an appropriate boundary condition. In the simulation, we consider CPU, Northbridge, Southbridge and DDR RAM chips as Two-Resistor simplified thermal models, each consisting of two parallelepiped components.

To set the boundary conditions for the inlet and outlet flows, we close the vents by placing a single lid on the inner side of each panel. Thus, we neglect some phenomena, which occur in the flow entering and exiting the case thought the vents. However, we take into account the value of the pressure loss coefficient reflecting the resistance to the flow in accordance with the specific shape and arrangement of the vent holes.



In the **SIMULATION_MODEL** instance you can see that the vents on the back panel are suppressed . This is done in order to define the exhaust fan boundary condition correctly. If you examine the original model geometry, you will see that the exhaust fan is placed close to the vents on the back panel, and there is no air flow through some of them. Actually, the air flow exits the case through a ring-shaped array of the vent holes (see the picture), so in the **SIMULATION_MODEL** instance we place a lid to close only these vent holes without considering other vent holes on the back panel at all. As resolving of each vent hole can be rather time-consuming and they are not the part of the flow simulation anyway, we suppress them. Instead, we specify an External Outlet Fan boundary condition on the inner surface of the ring-shaped lid. In addition, on the same lid we specify the Perforated Plate condition to define the pressure loss due to the resistance of the vent holes to the flow.

Creating a FloEFD Project

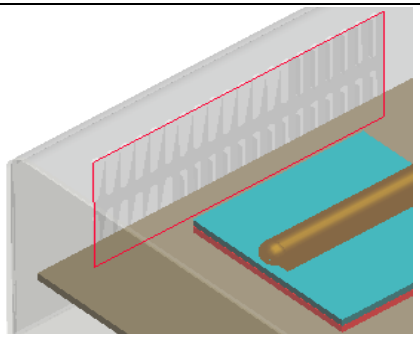
Using the **Wizard** create a new project as follows:

Project name	*Electronic components*
Configuration	*SIMULATION_MODEL*
Unit system	**SI**, *units for* **Temperature**: **°C***(***Celsius***)*
Analysis type	**Internal** **Exclude cavities without flow conditions**
Physical features	**Heat conduction in solids**, **Gravity**: **Y component** *of* `-9.81 m/s^2`
Default fluid	**Gases / Air**
Default solid	**Alloys / Steel (Mild)**
Wall Conditions	*Default outer wall thermal condition:* **Heat transfer coefficient** *of* `5.5 W/m^2/K`
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Default Result resolution level of 3;* *other options are default*

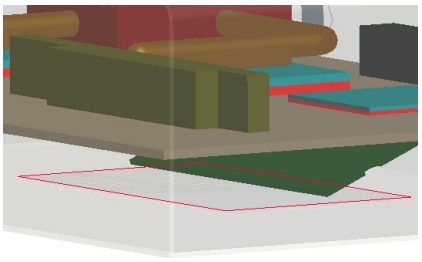
Specifying Boundary Conditions

Specify the boundary conditions for inlet and outlet flows as shown in the tables below:

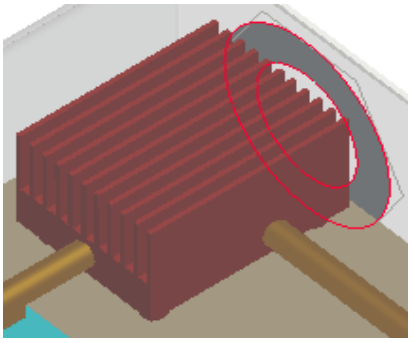
Type	**Environment Pressure**
Name	**Environment Pressure 1**
Faces to apply	the inner face of **INLET_LID**
Thermodynamic Parameters: Default values (101325 Pa and 20.05°C)	



Type	**Environment Pressure**
Name	**Environment Pressure 2**
Faces to apply	the inner face of the **INLET_LID_2**
Thermodynamic Parameters: Default values (101325 Pa and 20.05°C)	



Type	**External Outlet Fan**
Name	**External Outlet Fan 1**
Faces to apply	the inner face of the **OUTLET_LID**
Model: **Pre-Defined\Axial\Papst\Papst 412** **Thermodynamic Parameters**: Default values (101325 Pa)	



Specifying Perforated Plates

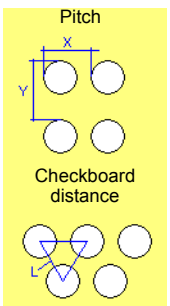
The **Perforated Plate** feature is used for simulating inlet and outlet flows through thin planar walls with multiple openings without having to create an individual lid for each opening. Instead, the **Perforated Plate** condition is applied together with a boundary condition for a surface of a single lid, which closes multiple openings, and defines the additional resistance of these openings to the flow. It can be useful, for example, when you simulate a flow entering or leaving the model through a series of small openings, which can require some additional mesh refinement if resolved directly. In this simulation, we use **Perforated Plates** to take into account the resistance of inlet and outlet vents in the computer case to the flow.

1 Click **Flow Analysis > Tools > Engineering Database**.

2 In the **Engineering Database**, under **Perforated Plates > User Defined**, create two items with the following parameters:

Property	Value
Name	Tutorial rectangular holes
Comments	
Hole shape	Rectangular
Height	0.01 m
Width	0.0015 m
Coverage	Pitch
X - pitch	0.003 m
Y - pitch	0.015 m

Property	Value
Name	Tutorial round holes
Comments	
Hole shape	Round
Diameter	0.003 m
Coverage	Checkerboard Distance
Distance between centers	0.004 m



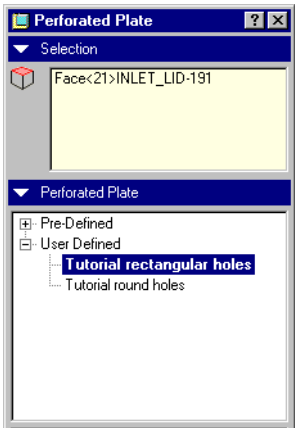
📖 *You can specify the **Hole shape** as **Rectangular > Round > Regular Polygon** or **Complex**. To define the holes arrangement (for non-**Complex** holes), in the **Coverage** you can select either **Pitch** or **Checkerboard distance** (for non-**Rectangular** holes). Depending on the selected option, you can specify the size of a single hole and either the distance between two adjacent holes in two mutually perpendicular directions (**X - Pitch** and **Y - Pitch**) or the **Distance between centers**. The specified values are used to calculate **Free area ratio**, which denotes the ratio of the holes total area to the total area of the perforated plate. The automatically calculated **Free area ratio** value appears at the bottom of the table. Alternatively, you can select the **Free area ratio** option in the **Coverage**, and specify this value directly.*

3 Save and exit the **Engineering Database**.

4 In the **Analysis tree**, select the **Environment Pressure 1** boundary condition.

5 Click **Flow Analysis > Insert > Perforated Plate**.

6 Under **Perforated Plate** select the created **Tutorial rectangular holes**.



7 Click **OK** . The new **Perforated Plate 1** item, which corresponds to the side vents of the case, appears in the **Analysis tree**.

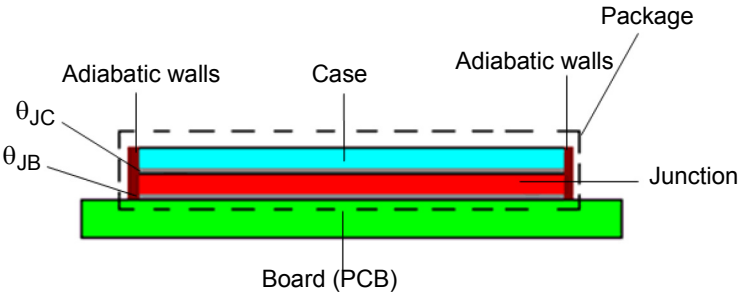
8 For the bottom vents, select the **Environment Pressure 2** boundary condition, and repeat steps 5-7.

9 For the back panel vents, select the **External Outlet Fan 1** condition, and repeat steps 5-7, selecting the **Tutorial round holes** item under **User Defined**.

📖 *The **Perforated Plate** feature is used in FloEFD simulation to define additional parameters for the already specified **Environment Pressure** or **Fan** conditions. It doesn't make any changes in the model geometry itself. So, when you delete the boundary condition or fan from your project, the corresponding **Perforated Plate** (if specified) becomes useless.*

Specifying Two-Resistor Components

The two-resistor model is widely used to estimate the temperature of chips and other small electronic packages. It considers each package consisting of two parallelepiped components (Case and Junction) with identical contact areas, made of material with high heat conductivity (see the picture below). The Junction-to-Case (θ_{JC}) and Junction-to-Board (θ_{JB}) thermal resistances are modeled as infinitely thin plates. The heat conduction through the package is calculated using the values of these resistances.



An extensive set of pre-defined two-resistor components is provided in the **Engineering Database**. Each item corresponds to a specific package type.

1 Click **Flow Analysis > Insert > Two-Resistor Component**

2 Select the **CPU_2R_CASE** component as **Case Body** and **CPU_2R_JUNCTION** as **Junction Body**.

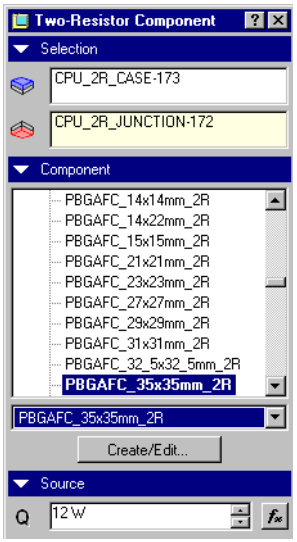
3 Under **Component** select the **PBGAFC_35x35mm_2R** item.

4 In the **Source**, enter the value of **Heat Generation Rate** Q equal to `12 W`.

5 Click **OK**. The new **Two-Resistor Component 1** item, which corresponds to CPU, appears in the **Analysis tree**.

6 Rename the created item to **CPU**. We will use this name for selecting this item to specify **Goals**.

7 In the same way specify the **Chipset – Northbridge** and **Chipset – Southbridge** items with the following parameters:



Name	Chipset - Northbridge
Case Body	NORTHBRIDGE_2R_CASE
Junction Body	NORTHBRIDGE_2R_JUNCTION
Component	PBGAFC_37_5x37_5mm_2R
Heat Generation Rate	`4.3 W`

Name	Chipset - Southbridge
Case Body	SOUTHBRIDGE_2R_CASE
Junction Body	SOUTHBRIDGE_2R_JUNCTION
Component	LQFP_256_28x28mm_2R
Heat Generation Rate	`2.5 W`

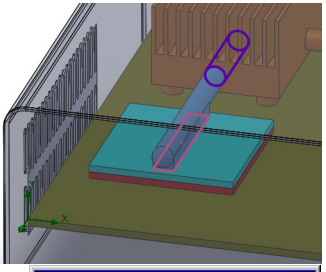
8 For each of the four considered DDR RAM chips, specify the same way **RAM chip N** item (with **N** being the chip number) by selecting its corresponding Case and Junction parts under the **SODIMM** assembly:

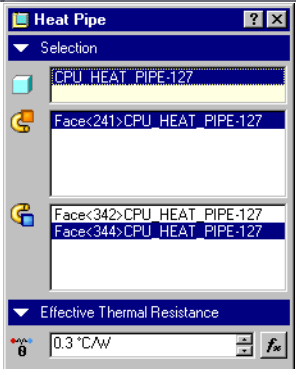
Name	RAM chip N
Case Body	RAM_CHIP_2R_CASE
Junction Body	RAM_CHIP_2R_JUNCTION
Component	TSOP_C_10_16x22_22_2R
Heat Generation Rate	1 W

📖 *If you specify some package as **Two-Resistor Component** in the project, make sure that its dimensions in the **Engineering Database** totally match (or are very close to) the dimensions of the package model geometry (its Case and Junction components). If the dimensions do not match, you must either make changes in the model geometry or select a different **Two-Resistor Component** in the **Engineering Database**.*

Specifying Heat Pipes

The **Heat Pipe** feature is used for modeling heat transfer from the hotter surface to the colder surface through a heat pipe (considered as solid body made of high heat-conducting material).





1 Click **Flow Analysis > Insert > Heat Pipe**.

2 Select **CPU_HEAT_PIPE** as **Components to Apply Heat Pipe**.

3 Select the face of the **CPU_HEAT_PIPE** component contacting with the top face of CPU as **Heat In Faces**.

4 Select the faces of the **CPU_HEAT_PIPE** contacting with the inner face of heat sink as **Heat Out Faces**.

5 Type the **Effective Thermal Resistance** value of `0.3 °C/W`. This value models the real efficiency of heat pipe.

6 Click **OK**. The new **Heat Pipe 1** item, which corresponds to the CPU heat pipe, appears in the **Analysis tree**.

7 In the same way specify the other heat pipe using the **NORTHBRIDGE_HEAT_PIPE** component with the same value of **Effective Thermal Resistance**.

Specifying Contact Resistances

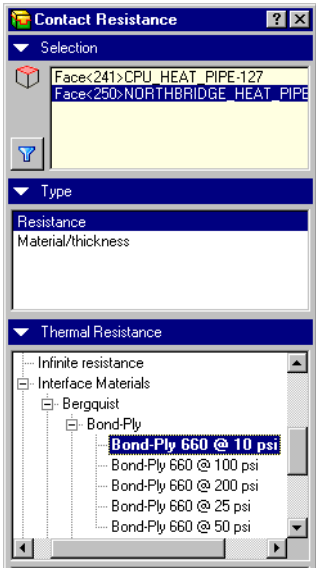
The **Contact Resistance** feature is used for specifying the value of thermal contact resistance on a face of a solid contacting fluid or another solid. It can be defined by a specific thermal resistance value or by thickness and thermal properties of the contact layer material. Taking into account the thermal contact resistance helps to estimate, for example, such phenomenon as temperature drop at the contact surface. Here we use this feature to specify thermal interface material attaching heat pipes to CPU and Northbridge and to specify thermal contact resistance between the surfaces of heat pipes and the surrounding air.

1 Click **Flow Analysis > Insert > Contact Resistance**.

2 Select the faces of the **CPU_HEAT_PIPE** and **NORTHBRIDGE_HEAT_PIPE** components contacting with the top faces of CPU and Northbridge correspondingly. We selected these faces earlier as **Heat In Faces** when specifying the heat pipes.

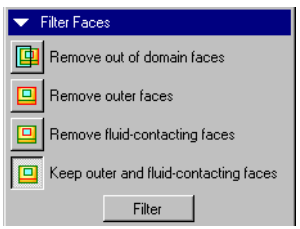
3 Under **Thermal Resistance,** select **Bond-Ply 660 @ 10 psi** (**Pre-Defined\Interface Materials\Bergquist\Bond-Ply\Bond-Ply 660 @ 10 psi**).

4 Click **OK** . The new **Contact Resistance 1** item appears in the **Analysis tree**.



5 Repeat step 1, then hold down the **Ctrl** key and double-click the **CPU_HEAT_PIPE** and **NORTHBRIDGE_HEAT_PIPE** components in the Model tree. FloEFD selects both these components.
Faces that are not in contact with fluid must be removed from the **Faces to Apply the Contact Resistance** list.

6 Under **Selection**, select each component in the selection list and click **List All Component Faces** in turn to list all faces belonging to the both components.



Then click **Filter Faces** . Select **Keep outer and fluid-contacting faces** , and click **Filter**.

🕮 *It is convenient to select all faces of the component by selecting this component in the Model Tree, though finding and removing unnecessary faces from the selection manually (one by one) may require excessive time, especially when there are many faces to remove. The **Filter** allows you to remove unnecessary faces of specified type from the list of selected faces.*

7 Under **Thermal Resistance**, expand the **Pre-Defined** list, and select **Infinite resistance**. We use **Infinite resistance** here to reflect the qualitative difference between the intensity of heat transfer inside and outside the considered heat pipes.

8 Click **OK** .

Specifying Printed Circuit Board

The **Printed Circuit Board** feature is used for modeling PCBs as flat solid bodies with anisotropic thermal conductivity, which is calculated from the specified structure of interleaving conductor and dielectric layers. You can define such material in the **Engineering Database** by specifying the properties of conductor and dielectric materials and the structure of layers. We use this feature to specify the material for SODIMM board, which consists of six layers of conductor (Copper) and five layers of dielectric (FR4)

1 Click **Flow Analysis > Tools > Engineering Database**.

2 In the **Engineering Database**, under **Printed Circuit Boards > User Defined**, create a new item with the following parameters:

Property	Value
Name	4s2p PCB
Comments	...
Type	Layer Definition
Dielectric material density	1200 kg/m^3
Dielectric material specific heat	880 J/(kg*K)
Dielectric material conductivity	0.3 W/(m*K)
Conductor material density	8960 kg/m^3
Conductor material specific heat	385 J/(kg*K)
Conductor material conductivity	401 W/(m*K)
PCB total thickness	0.001 m
Conducting layers	(Table) ...

🕮 *As you specify the parameters, at the bottom of this table you can see the calculated properties of the equivalent material used in the simulation .*

3 In the **Conducting Layers** table, click the ... button to switch to the **Tables and Curves** tab. Type the following values to specify the structure of conducting layers:

Layer Thickness	Percentage Cover
3.3e-005 m	20 %
6.6e-005 m	80 %
3.3e-005 m	20 %
3.3e-005 m	20 %
6.6e-005 m	80 %
3.3e-005 m	20 %

📖 *As you specify the layers structure, you can see the graphical representation of this structure at the right .*

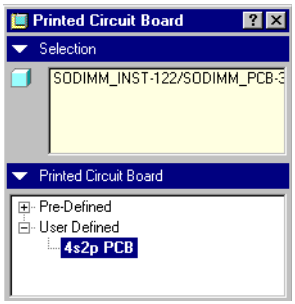
4 Save and exit the **Engineering Database**.

5 Click **Flow Analysis > Insert > Printed Circuit Board**.

6 Select **SODIMM_PCB** in the graphic area.

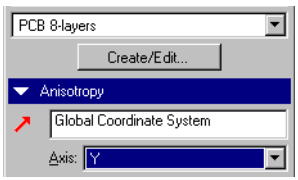
7 Under **Printed Circuit Board** select the created **4s2p PCB** item.

8 Click **OK** .



Specifying Solid Materials

For the **EPIC_PCB** component, we specify a non-isotropic material (**Pre-Defined\Non-isotropic\PCB 8-layers**) with Axisymmetrical/Biaxial conductivity. In this type of conductivity the thermal properties of the material are the same for two directions and differ for the third direction specified by an axis or direction.



To specify the axis for the **EPIC_PCB**, under **Anisotropy**, set **Axis** to **Y**.

For other components, specify the **Solid Material** as following:

Pre-Defined\Metals\Copper	**HEATSINK**
Pre-Defined\IC Packages\Typical Connector	**PC104_PCI_CONNECTOR , PC104_ISA_CONNECTOR, SODIMM_CONNECTOR**

To exclude the **INLET_LID**, **INLET_LID_2** and **OUTLET_LID** from the heat conduction analysis, specify them as insulators (**Pre-Defined\Glasses and Minerals\Insulator**).

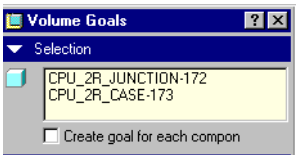
Specifying Project Goals

1 In the **Analysis tree**, select the two-resistor component **CPU**.

2 Click **Flow Analysis > Insert > Volume Goals**.

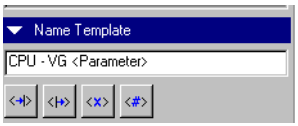
*In Creo Parametric click **Flow Analysis > Insert > Goals > Volume Goals**.*

3 In the **Components to Apply the Volume Goal** list, make sure that both Case (**CPU_2R_CASE**) and Junction (**CPU_2R_JUNCTION**) components are added.



4 Under **Parameter** select both **Max** and **Av Temperature (Solid)**.

5 Edit the **Name Template** to: `CPU - VG <Parameter>`.

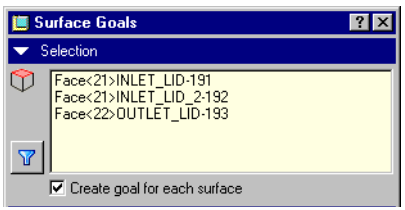


6 Click **OK** .

7 Repeat the same steps separately for each heat source: **Chipset - Northbridge**, **Chipset – Southbridge**, **RAM Chip 1**, **2**, **3**, **4** (select all these four RAM chips at once) and the **HEATSINK**. Edit the **Name Template** in a similar way.

8 When finished, in the **Analysis tree** select all specified boundary conditions (**Environment Pressure 1**, **Environment Pressure 2** and **External Outlet Fan 1**), holding down the **Ctrl** key.

9 Click **Flow Analysis > Insert > Surface Goals**.



*In Creo Parametric click **Flow Analysis > Insert > Goals > Surface Goals**.*

10 Select the **Separate goal for each surface** option to create a separate goal for each of the selected surfaces.

11 In the **Parameter**, select **Mass Flow Rate**.

12 Click **OK** .

Adjusting the Initial Mesh

1 Click **Flow Analysis > Project > Rebuild**.

1 To adjust the initial mesh settings, click **Flow Analysis > Initial Mesh**.

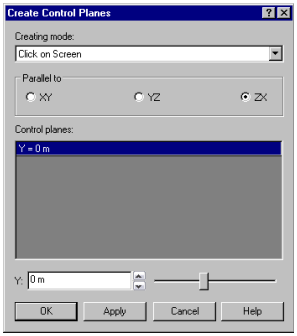
*In Creo Parametric click **Flow Analysis > Mesh > Initial Mesh**.*

2 Switch off the automatic settings by clearing the **Automatic settings** checkbox. Go to the **Basic Mesh** tab.

3 Click **Add Plane**.

4 In the **Create Control Planes** dialog box make sure that **Creating mode** is set to **Click on Screen**.
In the **Parallel to**, click **ZX**.

5 In the graphic area, click anywhere, and then type 0 for the **Y**.



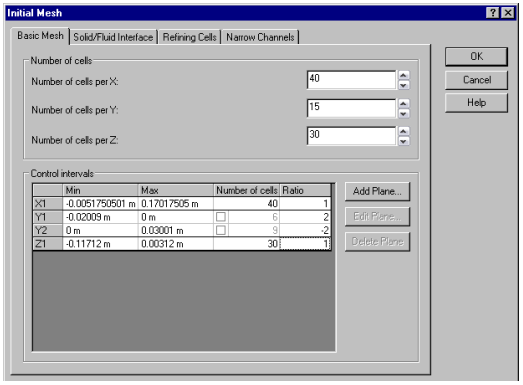
6 Click **OK** to return to the **Initial Mesh** window.

7 Click the **Ratio** cell of the **Y1** interval and enter the value of 2.

8 In the same manner enter the **Ratio** value of -2 for the **Y2** interval.

9 Specify the **Number of cells per X, Y and Z** of 40, 15, 30 respectively.

10 Go to the **Solid/Fluid Interface** tab and set **Small solid features refinement level** to 1.



11 Click **OK** to save initial mesh settings.

Specifying Local Initial Mesh Properties

It is also convenient to specify the **Local Initial Mesh** to obtain more accurate solution in the regions of interest.

1 In the **Analysis tree**, select all created Two-Resistor components (**CPU**, **Chipset – Northbridge**, **Chipset – Southbridge**, **RAM Chip 1-4**).

2 Click **Flow Analysis > Insert > Local Initial Mesh**.

*In Creo Parametric click **Flow Analysis > Mesh > Local Initial Mesh**.*

3 Clear the **Automatic settings** checkbox. Go to **Refining Cells** tab.

4 Select **Refine partial cells** and **Refine solid cells**.

5 Set both **Level of refining solid cells** and **Level of refining partial cells** to 2.

6 Click **OK** to save **Local Initial Mesh** settings.

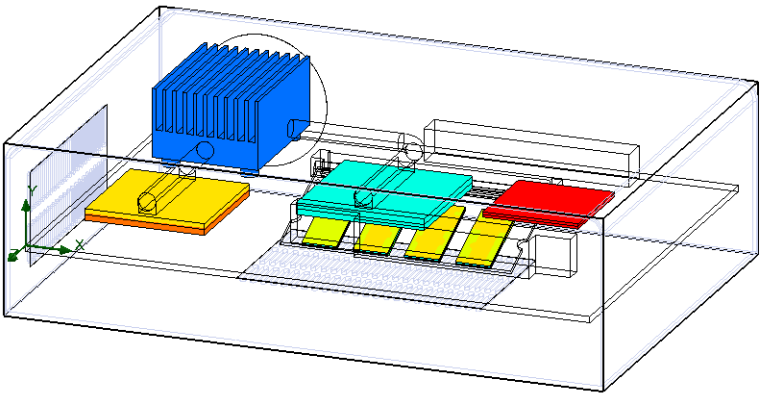
7 Create another **Local Initial Mesh** for the **HEATSINK**. After clearing the **Automatic settings** checkbox, go to the **Narrow Channels** tab, and set **Characteristic number of cells across a narrow channel** to 4 and **Narrow channels refinement level** to 2.

Save the model and run the calculation.

Results

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]
SG Mass Flow Rate Inlet Lid	[kg/s]	0,000874331	0,000872428	0,000866164	0,000876732	100
SG Mass Flow Rate Outlet Lid	[kg/s]	-0,002117667	-0,00211709	-0,002117667	-0,002116389	100
SG Mass Flow Rate Inlet Lid 2	[kg/s]	0,001243313	0,001244659	0,001240256	0,001250352	100
CPU - VG Av Temperature (Solid)	[°C]	78,11027219	78,09781048	78,05672621	78,11706281	100
CPU - VG Max Temperature (Solid)	[°C]	79,11610781	79,10387809	79,06256935	79,12328617	100
Chipset - Northbridge - VG Av Temperature (Solid)	[°C]	55,42071381	55,40562741	55,3841179	55,42256088	100
Chipset - Northbridge - VG Max Temperature (Solid)	[°C]	55,70526618	55,69034608	55,66890553	55,70711814	100
Chipset – Southbridge - VG Av Temperature (Solid)	[°C]	86,46932621	86,47201044	86,36744543	86,60810332	100
Chipset – Southbridge - VG Max Temperature (Solid)	[°C]	88,2380636	88,2376433	88,14342397	88,36032906	100
RAM Chips VG Av Temperature (Solid) 1	[°C]	63,75306739	63,70852983	63,65611637	63,75792045	100
RAM Chips VG Max Temperature (Solid) 1	[°C]	67,48810778	67,44384108	67,38708974	67,4944499	100
Heatsink VG Av Temperature (Solid) 1	[°C]	46,61653256	46,60195552	46,58783758	46,61751183	100
Heatsink VG Max Temperature (Solid) 1	[°C]	47,01342172	46,99872618	46,98476739	47,01442693	100

In accordance with the obtained results, we can say that electronic components operate at moderate temperatures, and there is no need to introduce any additional design features in order to improve the efficiency of heat exchange inside the considered case.



E2

LED Lighting

Some of the features used in this tutorial are available for the LED module users only.

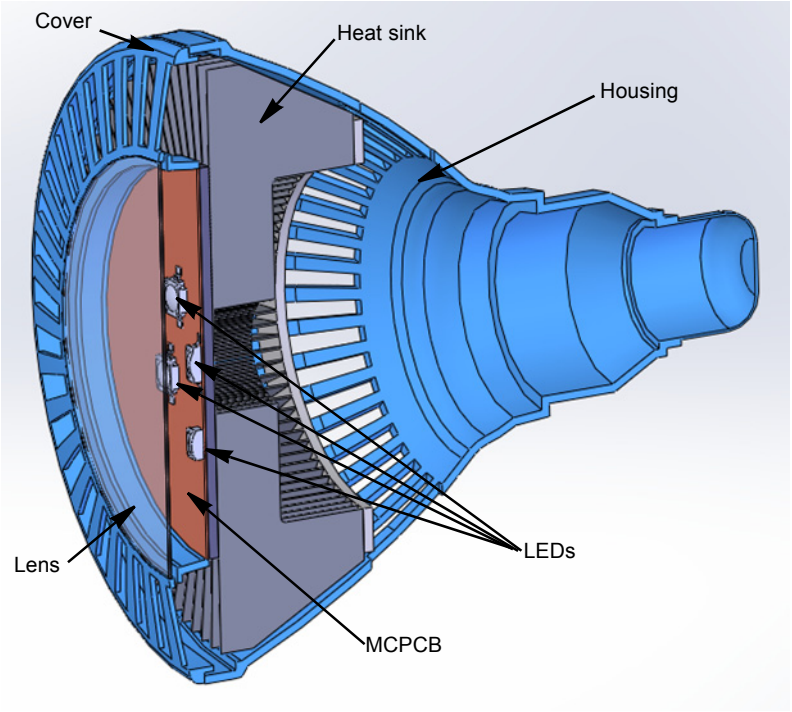
Problem Statement

The size, potential energy savings, and reduced maintenance benefits of LED's make them attractive for use in a wide range of applications. However, the thermal management is the most critical aspect of LED system design. Heat buildup can reduce an LED's light output and cause a color shift and at the same time. The key to successful LED system design is to transfer the active device's heat efficiently from its own PN junction to the ambient. The path involves both the printed circuit board that mounts the LED and the enclosure.

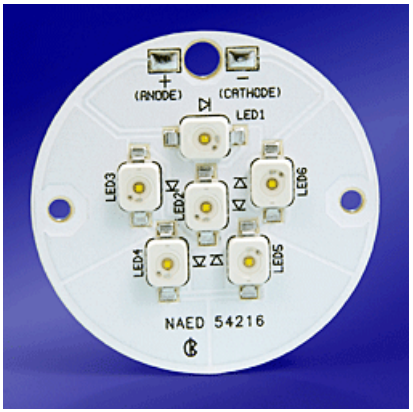
This tutorial demonstrates the capabilities of FloEFD to solve LED's thermal problems, helping you eliminate critical component temperatures, increase product reliability and ultimately product longevity.

The LED module features enable thermal simulation of the most advanced LED products along with the capability of predicting their "hot lumens" i.e. their total emitted luminous flux at their operating temperatures. This capability is provided through an LED model library whose elements are characterized directly from physical tests by the Mentor® T3Ster+TeraLED combined thermal and radiometric/photometric LED testing solution. In addition, the FloEFD LED models support radiation absorption in semi-transparent solids, such as the head or taillight lens.

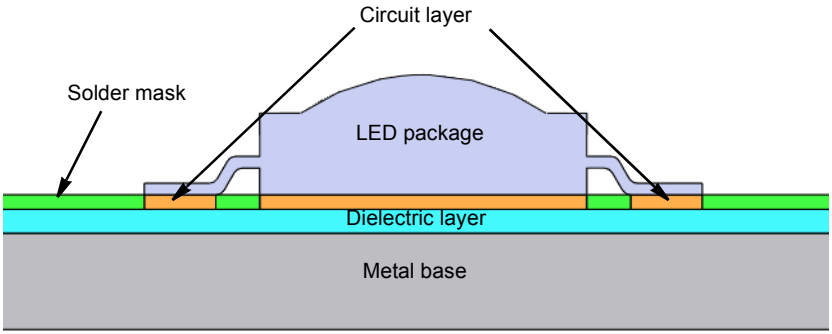
Here we consider a LED based lighting, which contains a six-LED array module, mounted on Aluminum heat sink and packaged in ABS polymer housing (including a cover), and a quartz glass lens as shown at the figure below.



The six-LED array module consists of six Osram Golden Dragon LEDs closely spaced on a low profile Metal Core Printed Circuit Board (MCPCB).



As Metal Core PCB means the base material for PCB is metal (usually aluminum, or copper alloy). MCPCBs are used instead of traditional PCBs because of the ability to efficiently dissipate heat away from the components. This is achieved by using a Thermally Conductive Dielectric Layer. Here we consider a single layer MCPCB consists of an aluminum base, T-preg dielectric layer, copper circuit layer and shellac solder mask. To simplify the model, we eliminate circuit layer and assume that LED packages is connected to the dielectric layer directly.



The lighting operates in typical indoor conditions at the room temperature (~20 °C) and without any forced cooling.

The objective of the simulation is to ensure that under these conditions, that lighting heat sink and housing participate efficiently in carrying heat away from the LEDs and the components of LED lighting are not overheated.

Opening the Model

Copy the **E2 - LED Lighting** folder into your working directory. Open the **LED_LIGHTING.asm** assembly. Select the generic instance.

📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **LED_LIGHTING.asm** assembly located in the **E2 - LED Lighting\Ready To Run** folder and run the project.*

Creating a FloEFD Project

Using the **Wizard** create a new project as follows:

Project name	*LED analysis*
Configuration	*Use current*
Unit system	**SI**
Analysis type	**External**
Physical features	**Heat conduction in solids**, **Radiation:** **Radiation model: Discrete Ordinates** **Environment radiation:** **Environment temperature** *of* `293.2 K`**;** **Absorption in solids;** **Gravity**: **Y component** of `-9.81 m/s^2`
Default fluid	**Gases / Air**
Default solid	**Alloys / Aluminium 5052**
Default wall conditions	**Default wall radiative surface: Pre-Defined / Real Surfaces / Aluminum, polished** *Default zero roughness*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	**Result resolution level** *of* `3` **Minimum gap size** *of* `0.02 m` **Minimum wall thickness** *of* `0.003 m` **Advanced narrow channel refinement***:* `On`

Adjusting the Computational Domain Size

Specify the computational domain size as follows:

X max = 0.4 m	Y max = 0.5 m	Z max = 0.4 m
X min = -0.4 m	Y min = -0.3 m	Z min = -0.4m

Specifying Solid Materials

For the opaque components, specify the **Solid Material** as following:

<table>
<tr><td>Cover
Housing</td><td>User Defined\ABS Polymer<table><tr><td>Density</td><td>890 kg/m^3</td></tr><tr><td>Specific heat</td><td>1900 J/(kg*K)</td></tr><tr><td>Conductivity type</td><td>Isotropic</td></tr><tr><td>Thermal conductivity</td><td>0.15 W/(m*K)</td></tr><tr><td>Electrical conductivity</td><td>Dielectric</td></tr><tr><td>Radiation properties</td><td>☐</td></tr><tr><td>Melting temperature</td><td>438.15 K</td></tr></table></td></tr>
<tr><td>MCPCB T-preg
(the upper layer of the Metal Core Printed Circuit Board)</td><td>User Defined\T-preg<table><tr><td>Density</td><td>1600 kg/m^3</td></tr><tr><td>Specific heat</td><td>1120 J/(kg*K)</td></tr><tr><td>Conductivity type</td><td>Isotropic</td></tr><tr><td>Thermal conductivity</td><td>2.2 W/(m*K)</td></tr><tr><td>Electrical conductivity</td><td>Dielectric</td></tr><tr><td>Radiation properties</td><td>☐</td></tr><tr><td>Melting temperature</td><td>438.15 K</td></tr></table></td></tr>
<tr><td>Heat Sink</td><td>Pre-Defined\Alloys\Aluminum 6061</td></tr>
</table>

Specifying Bodies and Materials Transparency

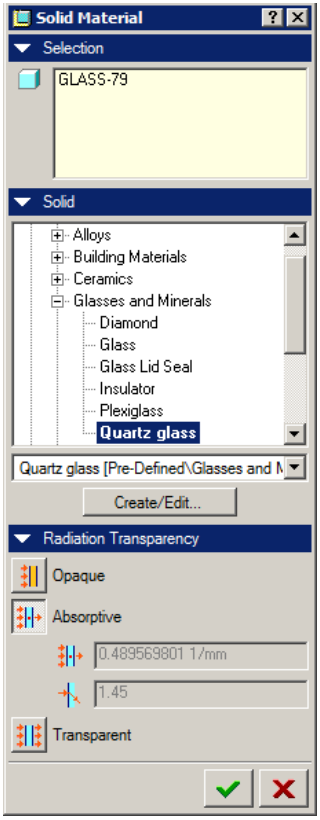
Assign the **Quartz glass** material to the lens and specify this component as semi-transparent to radiation.

1 Click **Flow Analysis > Insert > Solid Material**

2 In the Model Tree select the **GLASS** component.

3 Under **Solid** expand the list of **Pre-Defined** solid materials and select **Quartz Glass** under **Glasses and Minerals**.

4 Under **Radiation Transparency** select **Absorptive** .



📖 *Absorptive body is semi-transparent. It means that it absorbs the heat radiation within its volume. This option is available only if the absorption coefficient is specified in the solid material definition in the Engineering Database and the* ***Absorption in solids*** *check box is selected under* ***Radiation*** *in the* ***Wizard*** *or* ***General Settings****. The* ***Absorption Coefficient*** *and* ***Refractive Index*** *values are specified in the Engineering Database and are provided here just for reference.*

5 Click **OK** . FloEFD now treats this solid material and all solid bodies it is assigned to as semi-transparent to the thermal radiation.

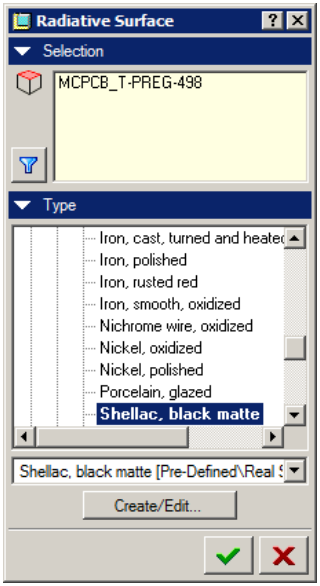
Specifying Radiative Surfaces

Follow the steps below to specify the radiative surfaces:

1 Click **Flow Analysis > Insert > Radiative Surface**.

1 In the graphics area select the outer face of the **MCPCB T-preg**.

1 Under **Type**, expand the list of **Pre-Defined\Real Surfaces** radiative surfaces and select **Shellac, black matter**.



2 Click **OK** .
Rename the new **Radiative Surface 1** item to `FR4 Radiative Surface`.

Click anywhere in the graphic area to clear the selection.

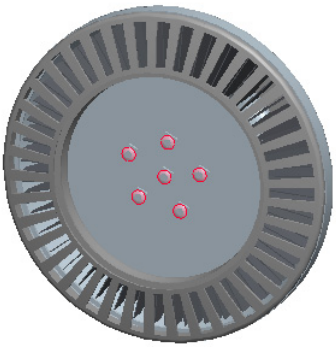
3 Click **Flow Analysis > Insert > Radiative Surface**.

4 Under **Selection**, select the top faces of LEDs.

5 Under **Type**, click **Create/Edit**.

6 In the **Engineering Database**, under **Radiative Surfaces > User Defined**, create a new item and change its **Name** to `avc-led`.

7 Change the parameters of the surface as shown below:



Radiative surface type	Wall
Specularity coefficient	0
Emissivity	Specific for thermal and solar radiation
Emissivity coefficient	0.7
Solar absorptance	0.7

8 Save the created radiative surface and exit the **Engineering Database**.

9 Under **Type**, expand the list of **User-Defined** radiative surfaces and select `avc-led`.

10 Click **OK** .
Change the name of the new radiative surface to `LED Radiative Surface`.

Specifying LEDs

The LED model implemented in FloEFD is based on a two-resistor model and a luminaire detailed model. The electric to light energy conversion efficiency is determined by the operating conditions of the LED: the junction temperature T_J and the applied forward current I_F. The power dissipated at the active region of a LED (P_H) is thus determined by the difference of the supplied total electrical energy $V_F \cdot I_F$ and the total energy emitted as optical radiation (Φ_e is emitted total radiant flux):

$$P_H = V_F \cdot I_F - \Phi_e$$

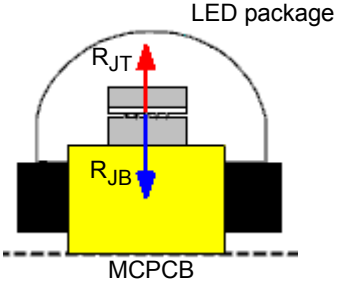
This power together with the LED thermal resistance R_{th} determine the junction temperature:

$$T_J = P_H \cdot R_{th} + T_{amb}$$

The total thermal resistance between the chip junction and the ambient can be given as follows:

$$\frac{1}{R_{th}} = \frac{1}{R_{JB}} + \frac{1}{R_{JT}}$$

where R_{JB} is the **Junction to bottom**, i.e. the total thermal resistance from the junction to the bottom side; R_{JT} is the **Junction to top**, i.e. the total thermal resistance between the junction and the topside of the chip ($R_{JT} >> R_{JB}$), by default it is set to $50 \cdot R_{JB}$.



The simple model based on a two-resistor model and the exact dissipated heating power P_H allows to predict the junction temperature T_J only.

The luminaire detailed model uses the exact forward current I_F and LED thermal characteristics and light output measured by T3Ster+TeraLED. This model allows to predict the junction temperature and the emitted luminous flux (Φ_V also called "hot lumens") and relies on the following assumptions:

- the forward voltage V_{F0} linearly depends on the junction temperature at fixed, constant forward current I_F :

$$V_F = V_{F0} - S_{VF} \cdot (T_J - T_0) \quad \textbf{(2.1)}$$

- the emitted optical power Φ_e and the emitted luminous flux Φ_V also linearly depends on the junction temperature at fixed, constant forward current I_F :

$$\Phi_e = \Phi_{e0} - S_{\Phi e} \cdot (T_J - T_0) \quad \textbf{(2.2)}$$

$$\Phi_V = \Phi_{V0} - S_{\Phi V} \cdot (T_J - T_0) \quad \textbf{(2.3)}$$

where:

S_{VF} is the temperature sensitivity of the forward voltage V_F at the constant forward current I_F ;

$S_{\Phi e}$ is the temperature sensitivity of the Φ_e radiant flux at the constant forward current I_F ;

$S_{\Phi V}$ is the temperature sensitivity of the Φ_V luminous flux at the constant forward current I_F ;

T_0 is the reference temperature.

Getting Data from T3Ster TeraLED

To conduct LED measurement it is recommended to follow the methods described in the JEDEC standards JESD51-51, JESD51-52 and JESD 51-14. The JESD51-14 document specifies a test method to determine the junction-to-case thermal resistance of semiconductor devices with a heat flow through a single path, such as LED. JESD51-51 and JESD51-52 provide general recommendations about thermal testing of LEDs. If you measure your LEDs with Mentor Graphics' T3Ster thermal transient test equipment in combination with the T3Ster TeraLED LED measurement environment and use the TeraLED Measurement Control software v2.0 (or higher) it is assured that your measurements will comply to these standards. (JESD51-51 defines the thermal test procedure for power LEDs, JESD51-52 provides recommendation about light output measurement of LEDs used in combination with thermal measurements.)

LED data you need to specify in FloEFD consists of an RC ladder model of that LED's heat-flow path (obtained from the step-wise approximation of the cumulative structure function), electrical and optical characteristics of the LED at forced, fixed forward current and at a given reference value of the LED's junction temperature (nominal value of the forward voltage and its temperature sensitivity, luminous flux and radiant flux and their temperature sensitivities).

You can get the dynamic thermal compact model (DCTM - or RC model) of the LED package from the T3Ster Master tool and the other data from the TeraLED View tool. If you characterize a single LED it is important to determine the RthJC value from a set of transient measurements as described in the JESD51-14 standard. The transient measurements used in this procedure must comply with the JESD51-51 and 51-52 standards as well (combined thermal and radiometric/photometric measurements). The RthJC value can be detected with the **Determine Rth-JC** command in T3Ster Master. After detecting RthJC you can export RC ladder to a ***.xCTM** file which you can import in FloEFD using **Import from T3Ster TeraLED** button in the Engineering database. This import specifies only RC ladder data, so the LED electrical and optical data should be taken from the TeraLED View. If you are not going to simulate transient analysis in FloEFD it is enough to input one RC pare only: RthJC as a resistance and the corresponding Capacitance value.

To get diode characteristics go to TeraLED View and open Diode Characteristics chart. Take voltage sensitivity (SVF) for the value of the forward current (I_{force}) for which you would like to create an LED model in FloEFD. Note that in FloEFD you need to input positive value of voltage sensitivity in SI system of units. When you set up the measurement of your LED in the TeraLED Measurement Control Tool (version 2.0 or higher) make sure that the chosen reference value of the junction temperature is among the pre-programmed temperature values. To get reference temperature, voltage ($U_{forward}$), radiant flux (P_{opt}), luminous flux (Φ), Radiant Flux Sensitivity ($S_{\varphi e}$) and Luminous Flux Sensitivity ($S_{\varphi v}$) go to **Optical Test Report** (you need version 2.0 or higher), select these data from the table of the test report which corresponds to your chosen reference temperature value. Convert the values into the SI system of units and change the Radiant Flux Sensitivity and Luminous Flux Sensitivity to positive, for example:

TeraLED View	FloEFD
$U_{forward}$ [V]	**Voltage [V]**
3.11	3.11
SVF [mV/°C]	**Voltage Sensitivity [V/K]**
-1.34	0.00134
P_{opt} [mW]	**Radiant Flux [W]**
313.3	0.3133
$S_{\varphi e}$ [mW/°C]	**Radiant Flux Sensitivity [W/K]**
-0.68	0.00068
Φ [lm]	**Luminous Flux [lm]**
93	93
$S_{\varphi v}$ [mlm/°C]	**Luminous Flux Sensitivity [lm/K]**
-227.6	0.2276

Specifying Pre-Defined LED components

A set of pre-defined LED components with thermal and optic characteristics measured by T3Ster+TeraLED is provided in the Engineering Database. Each item corresponds to a specific package type. The models of the pre-defined LED components are located in the **E2 - LED Lighting\Pre-Defined LEDs** folder.

Follow the steps below to specify the pre-defined LED components:

1 Click **Flow Analysis > Insert > LED**

2 Under **Selection**, select the central **OSRAM_GOLDEN_DRAGON** component as **Component to Apply the LED**, the lens face of this component as **Top Faces** and the bottom face of this component as **Bottom Faces**.

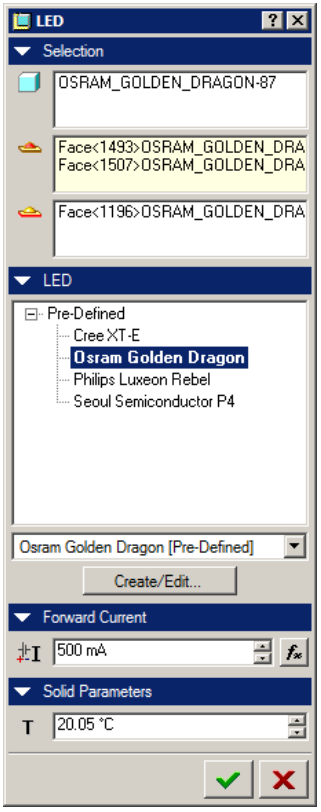
3 Under **LED** select the **Osram Golden Dragon** item.

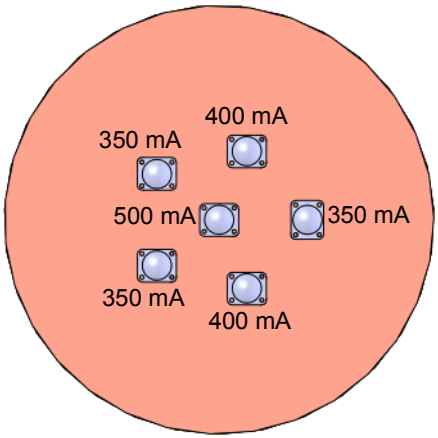
4 In the **Forward Current**, enter the value of **Current** equals to `500 mA`.

5 Click **OK**. The new **LED Component 1** item, which corresponds to the central LED, appears in the **Analysis tree**.

6 Rename the created item to `Osram Dragon LED 500 mA`. We will use this name for selecting this item to specify **Goals**.

7 In the same way specify the other five LED items with the current values shown below:

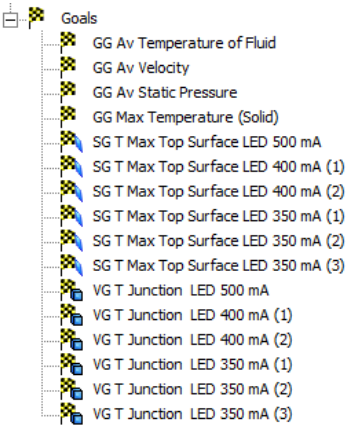




Specifying Goals

Specify surface goals of the maximum temperatures at the top (radiative) surfaces of the LEDs components and volume goals of the average temperatures at the LEDs components (you must select **Temperature (Solid)** as the goal parameter). In addition, you can specify global goals of average fluid parameters, such as velocity, temperature, static pressure, and the maximum solid temperature.
You can rename the goals as shown to make it easier to monitor them during the calculation.

Save the model and run the calculation.

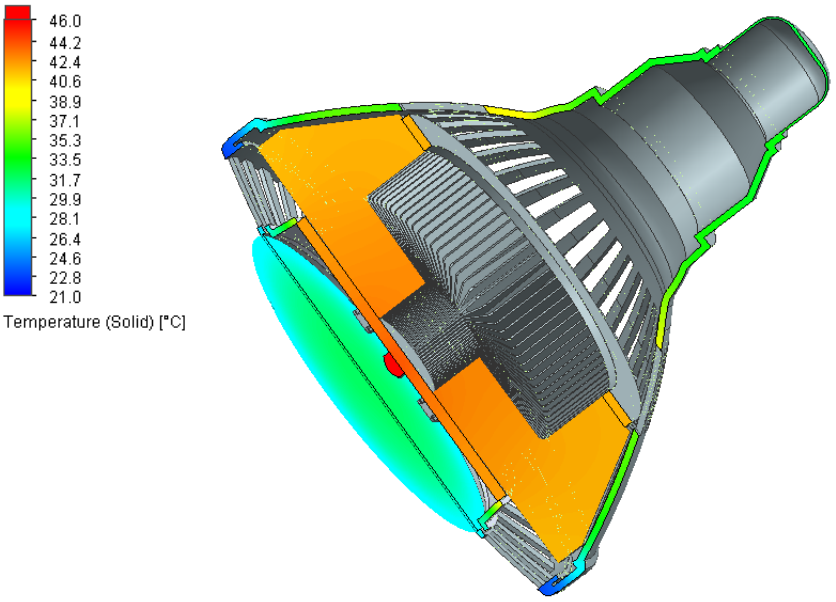

Goals
GG Av Temperature of Fluid
GG Av Velocity
GG Av Static Pressure
GG Max Temperature (Solid)
SG T Max Top Surface LED 500 mA
SG T Max Top Surface LED 400 mA (1)
SG T Max Top Surface LED 400 mA (2)
SG T Max Top Surface LED 350 mA (1)
SG T Max Top Surface LED 350 mA (2)
SG T Max Top Surface LED 350 mA (3)
VG T Junction LED 500 mA
VG T Junction LED 400 mA (1)
VG T Junction LED 400 mA (2)
VG T Junction LED 350 mA (1)
VG T Junction LED 350 mA (2)
VG T Junction LED 350 mA (3)

Results

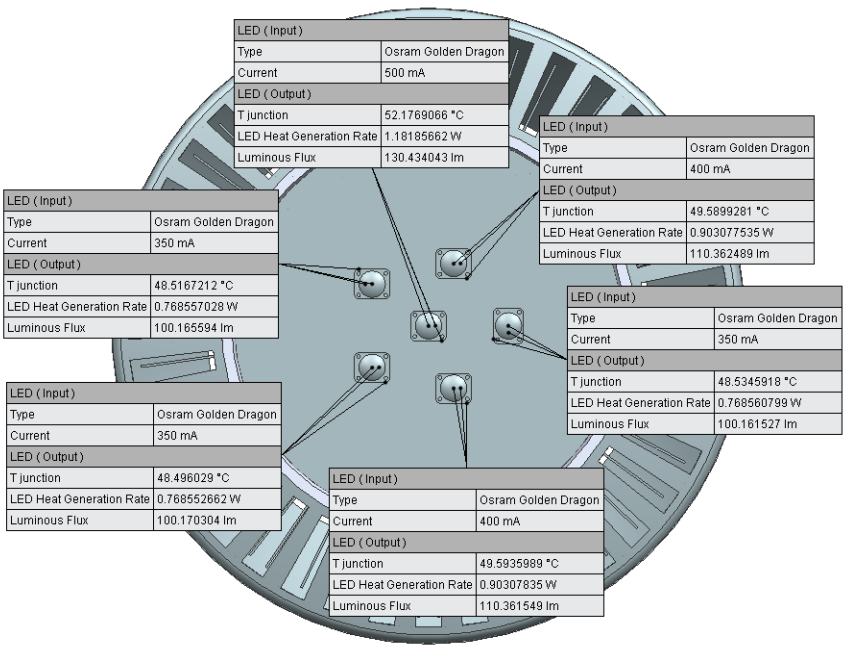
In accordance with the obtained results, we can say that LEDs operate at moderate temperatures, and there is no need to introduce any additional design features in order to improve the efficiency of heat exchange inside the considered housing.

Goal Name	Value, °C
SG T Max Top Surface LED 500 mA	45,2
SG T Max Top Surface LED 400 mA (1)	43,5
SG T Max Top Surface LED 400 mA (2)	44,0
SG T Max Top Surface LED 350 mA (1)	43,1
SG T Max Top Surface LED 350 mA (2)	43,6
SG T Max Top Surface LED 350 mA (3)	42,9
VG T Junction LED 500 mA	52,3
VG T Junction LED 400 mA (1)	49,7
VG T Junction LED 400 mA (2)	49,7
VG T Junction LED 350 mA (1)	48,6
VG T Junction LED 350 mA (2)	48,6
VG T Junction LED 350 mA (3)	48,6

Note that if you intend to display the distribution of the solid temperature, the Junction temperature fills the entire LED's section view.



To see the main LED characteristics, you can use callouts.


LED (Input)
Type Osram Golden Dragon
Current 500 mA
LED (Output)
T junction 52.1769066 °C
LED Heat Generation Rate 1.18185662 W
Luminous Flux 130.434043 lm
LED (Input)
Type Osram Golden Dragon
Current 400 mA
LED (Output)
T junction 49.5899281 °C
LED Heat Generation Rate 0.903077535 W
Luminous Flux 110.362489 lm
LED (Input)
Type Osram Golden Dragon
Current 350 mA
LED (Output)
T junction 48.5167212 °C
LED Heat Generation Rate 0.768557028 W
Luminous Flux 100.165594 lm
LED (Input)
Type Osram Golden Dragon
Current 350 mA
LED (Output)
T junction 48.5345918 °C
LED Heat Generation Rate 0.768560799 W
Luminous Flux 100.161527 lm
LED (Input)
Type Osram Golden Dragon
Current 350 mA
LED (Output)
T junction 48.496029 °C
LED Heat Generation Rate 0.768552662 W
Luminous Flux 100.170304 lm
LED (Input)
Type Osram Golden Dragon
Current 400 mA
LED (Output)
T junction 49.5935989 °C
LED Heat Generation Rate 0.90307835 W
Luminous Flux 110.361549 lm

F

Examples for Advanced CFD Module

The examples for **Advanced CFD Module** presented below demonstrate how to use capabilities and features of this module to solve specific engineering problems, such as the combustion of gas-phase mixtures. This functionality is available for the Advanced CFD module users only.

F1 - Combustion in a Tube

Examples for Advanced CFD Module:

F1

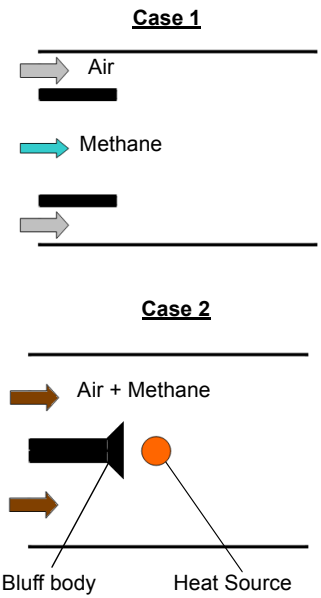
Combustion in a Tube

This feature is available for the Advanced CFD module users only.

Problem Statement

This tutorial demonstrates the capability of FloEFD to simulate combustion of gaseous mixture. It is shown how to define a project, i.e. specify the composition of fuel and oxidizer, boundary conditions and calculation goals.

As an example, let us perform the simulation of methane-air combustion in a cylindrical tube. Here, two cases are considered to demonstrate different approaches to the simulation of combustion in FloEFD. In the first case, methane (fuel) and air (oxidizer) are supplied into the tube separately and it is assumed that burning occurs infinitely fast as methane mixes with air. In the second case, methane and air are premixed and burn at a limited rate in the regions, where temperature of the mixture exceeds the specified ignition temperature. Comparing to the first case, there is also a bluff body in the center of the tube, which prevents the flame blowout and flashback. To heat the mixture above the ignition temperature, a local time-dependent heat source is used, which is placed in the wake of the bluff body.
Both cases are shown schematically in the figures at the right.



The considered tube has a length of 750 mm and a diameter of 50 mm. The inlet mass flow rates of methane and air are set to $5.5 \cdot 10^{-5}$ kg/s and $9.45 \cdot 10^{-4}$ kg/s, providing nearly stoichiometric mixture ratio. The inlet temperature is set to 293 K and the outlet pressure is 1 atm.

The objective of the simulation is to estimate thermal effects of the combustion in the tube.

Case 1: Non-premixed combustion

Opening the Model

Copy the **F1 - Combustion in Tube** folder into your working directory. Open the **Tube.asm** assembly. Select **The generic** instance.

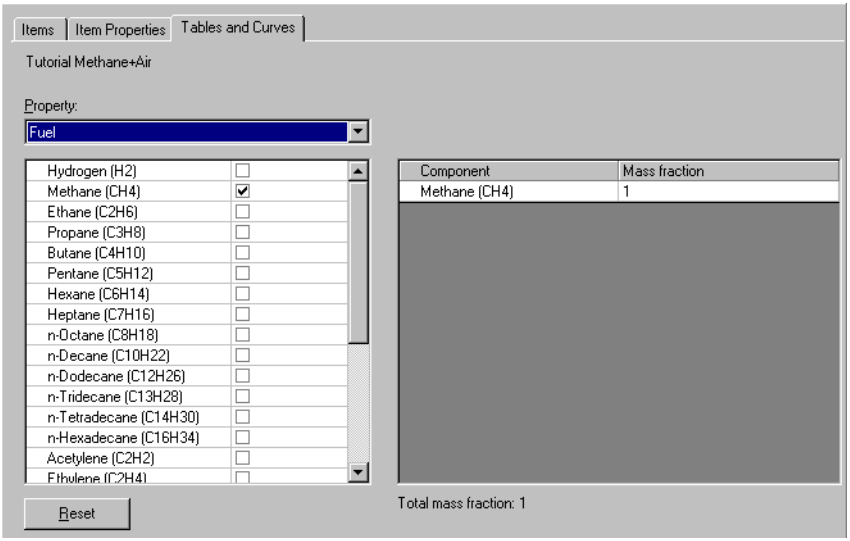
📖 *To skip the project definition and run the FloEFD project defined in accordance with the tutorial, you will need to open the **Tube.asm** assembly located in the **F1 - Combustion in Tube\Ready To Run** folder and run the desired projects.*

Defining Combustible Mixture in the Engineering Database

Before running the **Wizard**, specify the composition of the fuel and oxidizer in the **Combustible Mixture** used in this project.

1. Click **Flow Analysis > Tools > Engineering Database**.
2. In the **Engineering Database**, under **Materials > Combustible Mixtures > User Defined**, create a new item and change its **Name** to `Tutorial Methane+Air`.
3. Switch to the **Tables and Curves** tab and under **Property** select **Fuel**. In the list of available fuels, select **Methane (CH4)**.

4 In the table at the right, change the **Mass fraction** of **Methane (CH4)** to `1`.



5 Under **Property**, select **Oxidizer**. In the list of available oxidizers select **Air** and, similarly, change its **Mass fraction** to `1`.

🕮 *The **Total mass fraction** sums up the mass fractions of components separately for Fuel and Oxidizer. You must check that this value is equal to 1 for both the Fuel and the Oxidizer.*

6 Return to the **Item Properties** tab.

7 Change the other parameters of the mixture as shown below:

Minimum pressure	100000 Pa
Maximum pressure	110000 Pa
Minimum temperature	290 K
Maximum temperature	2500 K
Limited combustion rate	☐

🕮 *FloEFD calculates thermodynamic and thermophysical properties of a combustible mixture within the specified pressure and temperature ranges and stores them in a tabular form.*

8 Save the created combustible mixture and exit the **Engineering Database**.

Creating a FloEFD Project

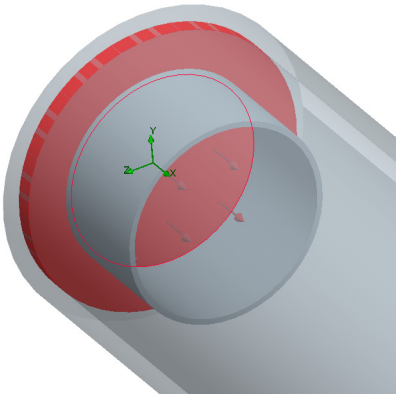
Using the **Wizard** create a new project as follows:

Project name	*Case 1 - Non-Premixed Combustion*
Unit system	**SI**
Analysis type	**Internal** **Exclude cavities without flow conditions**
Physical features	**Time-dependent** *(with default values)*
Default fluid	**Combustible Mixtures / Tutorial Methane+Air**
Wall Conditions	*Adiabatic wall with zero roughness*
Initial Conditions	*Default conditions*
Result and Geometry Resolution	*Default Result resolution level of 3;* *Minimum gap size = 0.015 m*

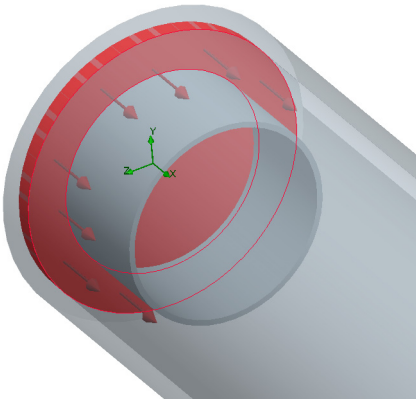
Specifying Boundary Conditions

Specify the boundary conditions for inlet and outlet flows as shown in the tables below:

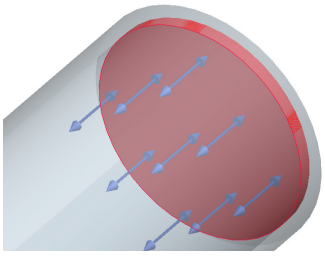
Type	**Inlet Mass Flow**
Name	**Methane Inlet Mass Flow**
Faces to apply	the inner circle-shaped face of the **INLET_LID1**
Mass Flow Rate of `5.5e-005 kg/s` **Thermodynamic Parameters**: Default values (101325 Pa and 293.2 K) **Substance Concentrations**: **Mass fraction** of **Fuel** is `1`; **Mass fraction** of **Oxidizer** is `0`	



Type	**Inlet Mass Flow**
Name	**Air Inlet Mass Flow**
Faces to apply	the inner ring-shaped face of the **INLET_LID1**
Mass Flow Rate of `9.45e-004 kg/s` **Thermodynamic Parameters**: Default values (101325 Pa and 293.2 K) **Substance Concentrations**: **Mass fraction** of **Fuel** is `0`; **Mass fraction** of **Oxidizer** is `1`	



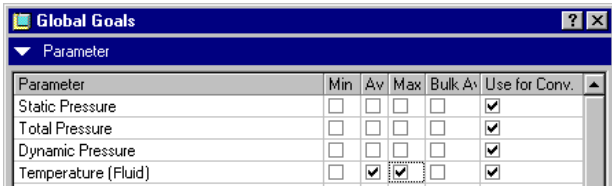
Type	**Environment Pressure**
Name	**Environment Pressure 1**
Faces to apply	the inner face of the **OUTLET_LID**
Thermodynamic Parameters: Default values (101325 Pa and 293.2 K)	



Specifying Project Goals

1 Click **Flow Analysis > Insert > Global Goals**.

*In Creo Parametric click **Flow Analysis > Insert > Goals > Global Goals**.*



2 Under **Parameter**, select both **Max** and **Av** for **Temperature (Fluid)**.

3 Click **OK** .

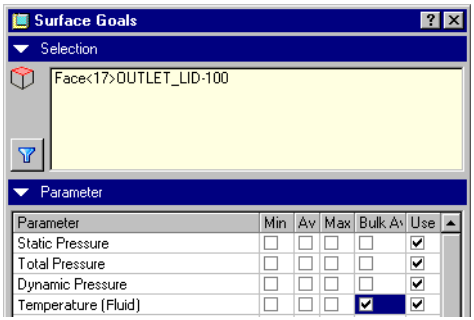
4 Click **Flow Analysis > Insert > Surface Goals**.

*In Creo Parametric click **Flow Analysis > Insert > Goals > Surface Goals**.*

5 In the **Analysis Tree**, select the **Environment Pressure 1** boundary condition.

6 Under **Parameter**, select **Bulk Av** for **Temperature (Fluid)**.

7 Click **OK** .

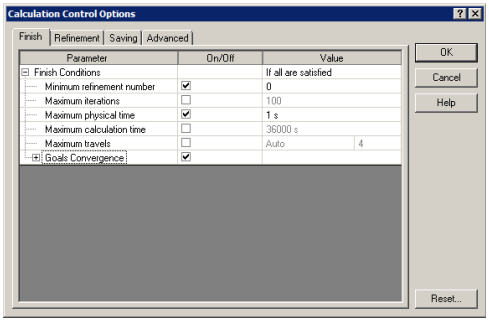


Adjusting the Calculation Control Options

1 Click **Flow Analysis > Calculation Control Options**.

*In Creo Parametric click **Flow Analysis > Project > Calculation Control Options**.*

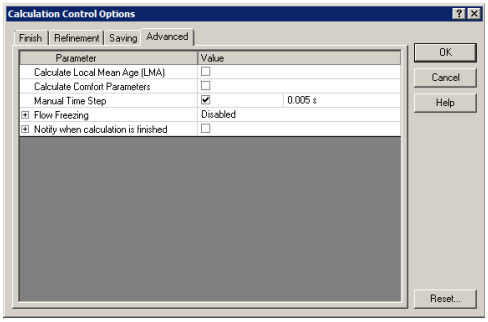
2 In the **Calculation Control Options** dialog box, change the **Value** of **Finish Conditions** to **If all are satisfied** and select **Goals Convergence**.



3 Switch to the **Advanced** tab.

4 Select the **Manual Time Step** check box and change its value to `0.005 s`. This value is appropriate for the given conditions and allows to obtain the solution sooner (the estimated default time step is smaller by more than an order of magnitude).

5 Click **OK**.

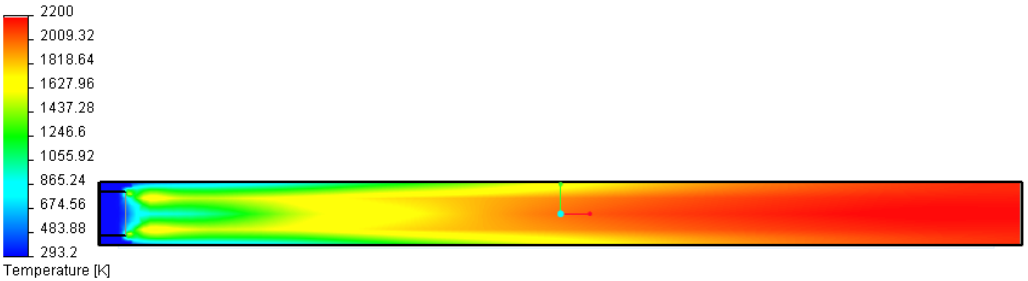


Save the model and run the calculation. If you are running this example for the first time and there is no calculated table with the thermodynamic and thermophysical properties of the **Tutorial Methane+Air** mixture, FloEFD calculates this table first.

🕮 *The calculated table of thermodynamic and thermophysical properties of a combustible mixture can be used in the future FloEFD projects involving this mixture. This table is not recalculated if there are no changes in the mixture properties (i.e. min/max values of temperature or pressure).*

Results

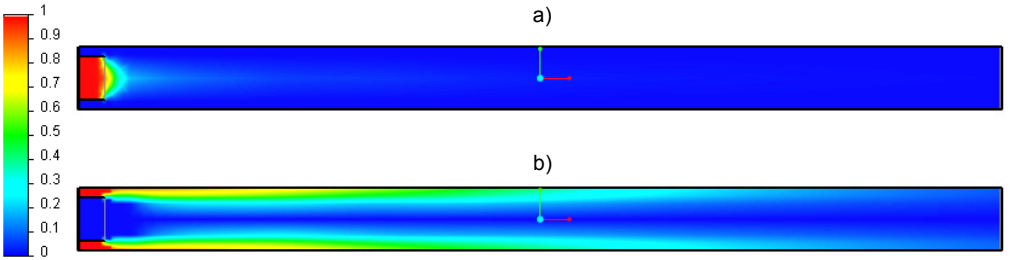
The distribution of fluid temperature along the tube is shown in the Cut Plot below. In this plot, the regions with the maximum temperature indicate the nearly stoichiometric mixture ratio.



Temperature distribution in the symmetry plane along the tube

Additionally, let us see the distribution of the Residual Fuel and the Residual Oxidizer along the tube.

To plot the **Mass Fraction of Residual Fuel** and **Mass Fraction of Residual Oxidizer** parameters, they must be enabled in the **Parameter** list (in the plot or Color Bar dialog you can add these parameters to the **Parameter** list by selecting **Add Parameter** at the end of the list).



The distribution of the Residual Fuel (a) and Residual Oxidizer (b) in the symmetry plane along the tube.

According to these plots, the fuel and the oxidizer burn out completely in the tube. The distribution of the unburned components shows the character of mixing along the tube.

It is also possible to output mass fractions of some specific combustion products. Let us see the mass fractions of the toxic combustion products, such as carbon oxides (CO_x) and nitrogen oxides (NO_x) calculated over the outlet surface. To display the **Mass Fraction of Carbon monoxide (CO)**, **Carbon dioxide (CO2)**, **Mass Fraction of Nitric oxide (NO)** and **Mass Fraction of Nitrogen dioxide (NO2)** parameters, they must be enabled in the **Parameter** list (in the plot or Color Bar dialog you can add these parameters to the **Parameter** list by selecting **Add Parameter** at the end of the list).

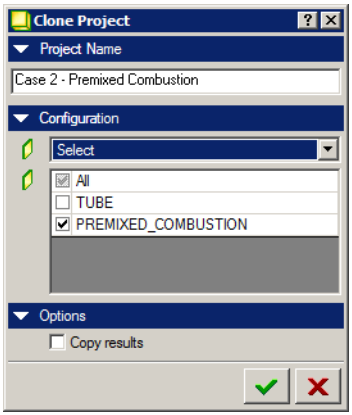
Parameter	Minimum	Maximum	Average	Bulk Average	Surface Area [m^2]
Mass Fraction of Carbon dioxide (CO2) [ ]	0,130038911	0,132320449	0,131201424	0,130948957	0,001944887
Mass Fraction of Carbon monoxide (CO) [ ]	0,001644667	0,012535915	0,004983696	0,006967012	0,001944887
Mass Fraction of Nitric oxide (NO) [ ]	0,002235115	0,003393448	0,003054349	0,002842074	0,001944887
Mass Fraction of Nitrogen dioxide (NO2) [ ]	1,2783E-06	2,70745E-06	2,178E-06	1,93511E-06	0,001944887

The mass fractions of the toxic components of the combustion products at the outlet surface.

Case 2: Premixed Combustion with ignition

Let us see how to solve a similar problem if the fuel and oxidizer are premixed. To calculate the combustion of premixed components, it is necessary to enable the **Limited Combustion Rate** option and specify the value of the **Ignition temperature** parameter in the **Tutorial Methane+Air** mixture. For this example, the value of ignition temperature is assumed equal to 700K. This means that combustion occurs only when the temperature of the mixture exceeds 700K. In order to provide the required temperature of ignition, a heat source of 100 W is placed downstream of the bluff body and is toggled on for 0.05 s.

1. Click **Flow Analysis > Project > Clone Project.**
2. In the **Project Name**, type `Case 2 - Premixed Combustion`
3. In the **Configuration to Add the Project** select **Select**.
4. In the **Configuration Name** list, select `PREMIXED_COMBUSTION`.
5. Click **OK** . When asked to reset the mesh settings, click **Yes**. Notice the warning, caused by the changes in geometry.
6. In the cloned project delete the **Methane Inlet Mass Flow** and **Air Inlet Mass Flow** boundary conditions.



7. Click **Flow Analysis > Tools > Engineering Database**.
8. Expand **Materials > Combustible Mixtures > User Defined** and copy/paste the **Tutorial Methane+Air** combustible mixture. Rename the new mixture to `Tutorial Methane+Air with Ignition` and modify its parameters as shown below:

Limited combustion rate	☑
Ignition temperature	700 K

9. Save the changes and exit the **Engineering Database**.
10. Click **Flow Analysis > General Settings** and under **Fluids** change the **Project fluids** to the recently created one (**Tutorial Methane+Air with Ignition**). When done, click **OK** to accept the changes.

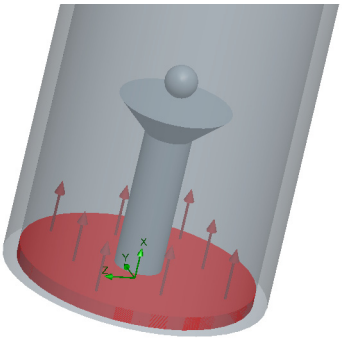
*To open **General Settings** dialog box in Creo Parametric, click **Flow Analysis > Project > General Settings**.*

11 On the ring face of the **INLET_LID**, create the **Inlet mass flow** boundary condition and set the values as shown below:

Flow Parameters:
Mass Flow Rate: 0.001 kg/s

Thermodynamic Parameters:
Default values (101325 Pa and 293 K)

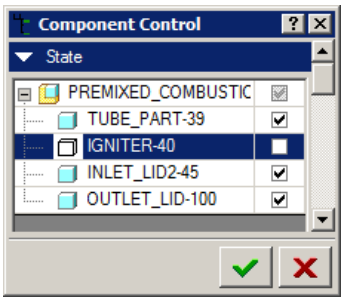
Substance Concentrations:
Mass Fraction of Fuel: 0.055;
Mass Fraction of Oxidizer: 0.945



12 Click **Flow Analysis > Component Control**.

In Creo Parametric click **Flow Analysis > Project > Component Control**.

13 In the **Component control** dialog box, deselect the **IGNITER** component, then click **OK**.
Now FloEFD will treat this component as a fluid.



14 Select the **IGNITER** component and specify the **Volume heat source** that is toggled on for the period of time from 0.15 s to 0.2 s. To do this, under **Toggle**, click **Dependency**.

15 Select **F(time)** as the **Dependency type** and fill the table as shown below. When finished, click **OK**.

Value t	Toggle
0.14 s	Off
0.15 s	On
0.2 s	Off

16 Specify the **Heat Generation Rate** of `100 W` and click **OK**.

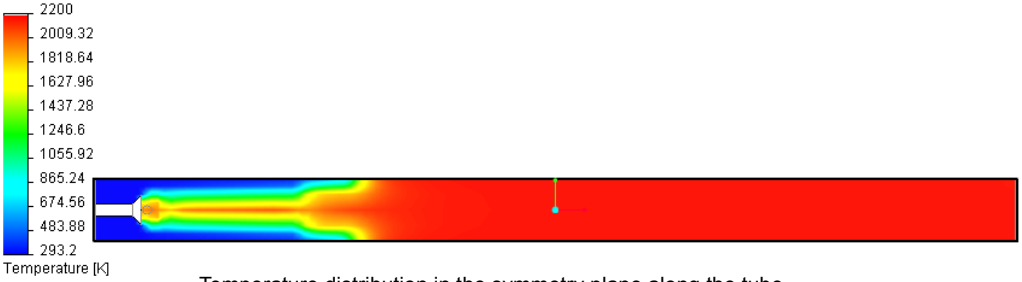
The power of the heat source required to ignite the combustible mixture depends on the volume of heat source, local temperature, heat capacity and velocity of the mixture.

There are other ways to heat the combustible mixture above the ignition temperature, such as supply of hot gases or presence of hot walls.

Save the model and run the calculation.

During the calculation, it is possible to track how the flow temperature changes in time. Notice that when the maximum temperature of the mixture is below the specified ignition temperature, mixture does not burn. After 0.15 s, the heat source toggles on, heats the mixture above the ignition temperature, and the burning starts.

The Cut Plot of the calculated temperature is shown below.



Temperature distribution in the symmetry plane along the tube

Since the given methane/air mixture ratio is nearly stoichometric, the maximum temperature is the same as in the previously calculated project.

End-User License Agreement

The latest version of the End-User License Agreement is available on-line at:
www.mentor.com/eula

IIMPORTANT INFORMATION

USE OF ALL SOFTWARE IS SUBJECT TO LICENSE RESTRICTIONS. CAREFULLY READ THIS LICENSE AGREEMENT BEFORE USING THE PRODUCTS. USE OF SOFTWARE INDICATES CUSTOMER'S COMPLETE AND UNCONDITIONAL ACCEPTANCE OF THE TERMS AND CONDITIONS SET FORTH IN THIS AGREEMENT. ANY ADDITIONAL OR DIFFERENT PURCHASE ORDER TERMS AND CONDITIONS SHALL NOT APPLY.

END-USER LICENSE AGREEMENT ("Agreement")

This is a legal agreement concerning the use of Software (as defined in Section 2) and hardware (collectively "Products") between the company acquiring the Products ("Customer"), and the Mentor Graphics entity that issued the corresponding quotation or, if no quotation was issued, the applicable local Mentor Graphics entity ("Mentor Graphics"). Except for license agreements related to the subject matter of this license agreement which are physically signed by Customer and an authorized representative of Mentor Graphics, this Agreement and the applicable quotation contain the parties' entire understanding relating to the subject matter and supersede all prior or contemporaneous agreements. If Customer does not agree to these terms and conditions, promptly return or, in the case of Software received electronically, certify destruction of Software and all accompanying items within five days after receipt of Software and receive a full refund of any license fee paid.

1. **ORDERS, FEES AND PAYMENT.**

   1.1. To the extent Customer (or if agreed by Mentor Graphics, Customer's appointed third party buying agent) places and Mentor Graphics accepts purchase orders pursuant to this Agreement ("Order(s)"), each Order will constitute a contract between Customer and Mentor Graphics, which shall be governed solely and exclusively by the terms and conditions of this Agreement, any applicable addenda and the applicable quotation, whether or not these documents are referenced on the Order. Any additional or conflicting terms and conditions appearing on an Order or presented via any electronic portal or other automated order management system will not be effective unless agreed in writing by an authorized representative of Customer and Mentor Graphics.

   1.2. Amounts invoiced will be paid, in the currency specified on the applicable invoice, within 30 days from the date of such invoice. Any past due invoices will be subject to the imposition of interest charges in the amount of one and one-half percent per month or the applicable legal rate currently in effect, whichever is lower. Prices do not include freight, insurance, customs duties, taxes or other similar charges, which Mentor Graphics will state separately in the applicable invoice(s). Unless timely provided with a valid certificate of exemption or other evidence that items are not taxable, Mentor Graphics will invoice Customer for all applicable taxes including, but not limited to, VAT, GST, sales tax, consumption tax and service tax. Customer will make all payments free and clear of, and without reduction for, any withholding or other taxes; any such taxes imposed on payments by Customer hereunder will be Customer's sole responsibility. If Customer appoints a third party to place purchase orders and/or make payments on Customer's behalf, Customer shall be liable for payment under Orders placed by such third party in the event of default.

1.3. All Products are delivered FCA factory (Incoterms 2010), freight prepaid and invoiced to Customer, except Software delivered electronically, which shall be deemed delivered when made available to Customer for download. Mentor Graphics retains a security interest in all Products delivered under this Agreement, to secure payment of the purchase price of such Products, and Customer agrees to sign any documents that Mentor Graphics determines to be necessary or convenient for use in filing or perfecting such security interest. Mentor Graphics' delivery of Software by electronic means is subject to Customer's provision of both a primary and an alternate e-mail address.

2. **GRANT OF LICENSE.** The software installed, downloaded, or otherwise acquired by Customer under this Agreement, including any updates, modifications, revisions, copies, documentation and design data ("Software") are copyrighted, trade secret and confidential information of Mentor Graphics or its licensors, who maintain exclusive title to all Software and retain all rights not expressly granted by this Agreement. Mentor Graphics grants to Customer, subject to payment of applicable license fees, a nontransferable, nonexclusive license to use Software solely: (a) in machine-readable, object-code form (except as provided in Subsection 5.2); (b) for Customer's internal business purposes; (c) for the term of the license; and (d) on the computer hardware and at the site authorized by Mentor Graphics. A site is restricted to a one-half mile (800 meter) radius. Customer may have Software temporarily used by an employee for telecommuting purposes from locations other than a Customer office, such as the employee's residence, an airport or hotel, provided that such employee's primary place of employment is the site where the Software is authorized for use. Mentor Graphics' standard policies and programs, which vary depending on Software, license fees paid or services purchased, apply to the following: (a) relocation of Software; (b) use of Software, which may be limited, for example, to execution of a single session by a single user on the authorized hardware or for a restricted period of time (such limitations may be technically implemented through the use of authorization codes or similar devices); and (c) support services provided, including eligibility to receive telephone support, updates, modifications, and revisions. For the avoidance of doubt, if Customer provides any feedback or requests any change or enhancement to Products, whether in the course of receiving support or consulting services, evaluating Products, performing beta testing or otherwise, any inventions, product improvements, modifications or developments made by Mentor Graphics (at Mentor Graphics' sole discretion) will be the exclusive property of Mentor Graphics.

3. **ESC SOFTWARE.** If Customer purchases a license to use development or prototyping tools of Mentor Graphics' Embedded Software Channel ("ESC"), Mentor Graphics grants to Customer a nontransferable, nonexclusive license to reproduce and distribute executable files created using ESC compilers, including the ESC run-time libraries distributed with ESC C and C++ compiler Software that are linked into a composite program as an integral part of Customer's compiled computer program, provided that Customer distributes these files only in conjunction with Customer's compiled computer program. Mentor Graphics does NOT grant Customer any right to duplicate, incorporate or embed copies of Mentor Graphics' real-time operating systems or other embedded software products into Customer's products or applications without first signing or otherwise agreeing to a separate agreement with Mentor Graphics for such purpose.

4. **BETA CODE.**

4.1. Portions or all of certain Software may contain code for experimental testing and evaluation (which may be either alpha or beta, collectively "Beta Code"), which may not be used without Mentor Graphics' explicit authorization. Upon Mentor Graphics' authorization, Mentor Graphics grants to Customer a temporary, nontransferable, nonexclusive license for experimental use to test and evaluate the Beta Code without charge for a limited period of time specified by Mentor Graphics. This grant and Customer's use of the Beta Code shall not be construed as marketing or offering to sell a license to the Beta Code, which Mentor Graphics may choose not to release commercially in any form.

4.2. If Mentor Graphics authorizes Customer to use the Beta Code, Customer agrees to evaluate and test the Beta Code under normal conditions as directed by Mentor Graphics. Customer will contact Mentor Graphics periodically during Customer's use of the Beta Code to discuss any malfunctions or suggested improvements. Upon completion of Customer's evaluation and testing, Customer will send to Mentor Graphics a written evaluation of the Beta Code, including its strengths, weaknesses and recommended improvements.

4.3. Customer agrees to maintain Beta Code in confidence and shall restrict access to the Beta Code, including the methods and concepts utilized therein, solely to those employees and Customer location(s) authorized by Mentor Graphics to perform beta testing. Customer agrees that any written evaluations and all inventions, product improvements, modifications or developments that Mentor Graphics conceived or made during or subsequent to this Agreement, including those based partly or wholly on Customer's feedback, will be the exclusive property of Mentor Graphics. Mentor Graphics will have exclusive rights, title and interest in all such property. The provisions of this Subsection 4.3 shall survive termination of this Agreement.

5. **RESTRICTIONS ON USE.**

5.1. Customer may copy Software only as reasonably necessary to support the authorized use. Each copy must include all notices and legends embedded in Software and affixed to its medium and container as received from Mentor Graphics. All copies shall remain the property of Mentor Graphics or its licensors. Customer shall maintain a record of the number and primary location of all copies of Software, including copies merged with other software, and shall make those records available to Mentor Graphics upon request. Customer shall not make Products available in any form to any person other than Customer's employees and on-site contractors, excluding Mentor Graphics competitors, whose job performance requires access and who are under obligations of confidentiality. Customer shall take appropriate action to protect the confidentiality of Products and ensure that any person permitted access does not disclose or use Products except as permitted by this Agreement. Customer shall give Mentor Graphics written notice of any unauthorized disclosure or use of the Products as soon as Customer becomes aware of such unauthorized disclosure or use. Except as otherwise permitted for purposes of interoperability as specified by applicable and mandatory local law, Customer shall not reverse-assemble, reverse-compile, reverse-engineer or in any way derive any source code from Software. Log files, data files, rule files and script files generated by or for the Software (collectively "Files"), including without limitation files containing Standard Verification Rule Format ("SVRF") and Tcl Verification Format ("TVF") which are Mentor Graphics' proprietary syntaxes for expressing process rules, constitute or include confidential information of Mentor Graphics. Customer may share Files with third parties, excluding Mentor Graphics competitors, provided that the confidentiality of such Files is protected by written agreement at least as well as Customer protects other information of a similar nature or importance, but in any case with at least reasonable care. Customer may use Files containing SVRF or TVF only with Mentor Graphics products. Under no circumstances shall Customer use Software or Files or allow their use for the purpose of developing, enhancing or marketing any product that is in any way competitive with Software, or disclose to any third party the results of, or information pertaining to, any benchmark.

5.2. If any Software or portions thereof are provided in source code form, Customer will use the source code only to correct software errors and enhance or modify the Software for the authorized use. Customer shall not disclose or permit disclosure of source code, in whole or in part, including any of its methods or concepts, to anyone except Customer's employees or on-site contractors, excluding Mentor Graphics competitors, with a need to know. Customer shall not copy or compile source code in any manner except to support this authorized use.

5.3. Customer may not assign this Agreement or the rights and duties under it, or relocate, sublicense or otherwise transfer the Products, whether by operation of law or otherwise ("Attempted Transfer"), without Mentor Graphics' prior written consent and payment of

Mentor Graphics' then-current applicable relocation and/or transfer fees. Any Attempted Transfer without Mentor Graphics' prior written consent shall be a material breach of this Agreement and may, at Mentor Graphics' option, result in the immediate termination of the Agreement and/or the licenses granted under this Agreement. The terms of this Agreement, including without limitation the licensing and assignment provisions, shall be binding upon Customer's permitted successors in interest and assigns.

5.4. The provisions of this Section 5 shall survive the termination of this Agreement.

6. **SUPPORT SERVICES.** To the extent Customer purchases support services, Mentor Graphics will provide Customer with updates and technical support for the Products, at the Customer site(s) for which support is purchased, in accordance with Mentor Graphics' then current End-User Support Terms located at http://supportnet.mentor.com/about/legal/.

7. **LIMITED WARRANTY.**

7.1. Mentor Graphics warrants that during the warranty period its standard, generally supported Products, when properly installed, will substantially conform to the functional specifications set forth in the applicable user manual. Mentor Graphics does not warrant that Products will meet Customer's requirements or that operation of Products will be uninterrupted or error free. The warranty period is 90 days starting on the 15th day after delivery or upon installation, whichever first occurs. Customer must notify Mentor Graphics in writing of any nonconformity within the warranty period. For the avoidance of doubt, this warranty applies only to the initial shipment of Software under an Order and does not renew or reset, for example, with the delivery of (a) Software updates or (b) authorization codes or alternate Software under a transaction involving Software re-mix. This warranty shall not be valid if Products have been subject to misuse, unauthorized modification, improper installation or Customer is not in compliance with this Agreement. MENTOR GRAPHICS' ENTIRE LIABILITY AND CUSTOMER'S EXCLUSIVE REMEDY SHALL BE, AT MENTOR GRAPHICS' OPTION, EITHER (A) REFUND OF THE PRICE PAID UPON RETURN OF THE PRODUCTS TO MENTOR GRAPHICS OR (B) MODIFICATION OR REPLACEMENT OF THE PRODUCTS THAT DO NOT MEET THIS LIMITED WARRANTY. MENTOR GRAPHICS MAKES NO WARRANTIES WITH RESPECT TO: (A) SERVICES; (B) PRODUCTS PROVIDED AT NO CHARGE; OR (C) BETA CODE; ALL OF WHICH ARE PROVIDED "AS IS."

7.2. THE WARRANTIES SET FORTH IN THIS SECTION 7 ARE EXCLUSIVE. NEITHER MENTOR GRAPHICS NOR ITS LICENSORS MAKE ANY OTHER WARRANTIES EXPRESS, IMPLIED OR STATUTORY, WITH RESPECT TO PRODUCTS PROVIDED UNDER THIS AGREEMENT. MENTOR GRAPHICS AND ITS LICENSORS SPECIFICALLY DISCLAIM ALL IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NON-INFRINGEMENT OF INTELLECTUAL PROPERTY.

8. **LIMITATION OF LIABILITY.** EXCEPT WHERE THIS EXCLUSION OR RESTRICTION OF LIABILITY WOULD BE VOID OR INEFFECTIVE UNDER APPLICABLE LAW, IN NO EVENT SHALL MENTOR GRAPHICS OR ITS LICENSORS BE LIABLE FOR INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES (INCLUDING LOST PROFITS OR SAVINGS) WHETHER BASED ON CONTRACT, TORT OR ANY OTHER LEGAL THEORY, EVEN IF MENTOR GRAPHICS OR ITS LICENSORS HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN NO EVENT SHALL MENTOR GRAPHICS' OR ITS LICENSORS' LIABILITY UNDER THIS AGREEMENT EXCEED THE AMOUNT RECEIVED FROM CUSTOMER FOR THE HARDWARE, SOFTWARE LICENSE OR SERVICE GIVING RISE TO THE CLAIM. IN THE CASE WHERE NO AMOUNT WAS PAID, MENTOR GRAPHICS AND ITS LICENSORS SHALL HAVE NO LIABILITY FOR ANY DAMAGES WHATSOEVER. THE PROVISIONS OF THIS SECTION 8 SHALL SURVIVE

THE TERMINATION OF THIS AGREEMENT.

9. **HAZARDOUS APPLICATIONS.** CUSTOMER ACKNOWLEDGES IT IS SOLELY RESPONSIBLE FOR TESTING ITS PRODUCTS USED IN APPLICATIONS WHERE THE FAILURE OR INACCURACY OF ITS PRODUCTS MIGHT RESULT IN DEATH OR PERSONAL INJURY ("HAZARDOUS APPLICATIONS"). EXCEPT TO THE EXTENT THIS EXCLUSION OR RESTRICTION OF LIABILITY WOULD BE VOID OR INEFFECTIVE UNDER APPLICABLE LAW, IN NO EVENT SHALL MENTOR GRAPHICS OR ITS LICENSORS BE LIABLE FOR ANY DAMAGES RESULTING FROM OR IN CONNECTION WITH THE USE OF MENTOR GRAPHICS PRODUCTS IN OR FOR HAZARDOUS APPLICATIONS. THE PROVISIONS OF THIS SECTION 9 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

10. **INDEMNIFICATION.** CUSTOMER AGREES TO INDEMNIFY AND HOLD HARMLESS MENTOR GRAPHICS AND ITS LICENSORS FROM ANY CLAIMS, LOSS, COST, DAMAGE, EXPENSE OR LIABILITY, INCLUDING ATTORNEYS' FEES, ARISING OUT OF OR IN CONNECTION WITH THE USE OF MENTOR GRAPHICS PRODUCTS IN OR FOR HAZARDOUS APPLICATIONS. THE PROVISIONS OF THIS SECTION 10 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

11. **INFRINGEMENT.**

    11.1. Mentor Graphics will defend or settle, at its option and expense, any action brought against Customer in the United States, Canada, Japan, or member state of the European Union which alleges that any standard, generally supported Product acquired by Customer hereunder infringes a patent or copyright or misappropriates a trade secret in such jurisdiction. Mentor Graphics will pay costs and damages finally awarded against Customer that are attributable to such action. Customer understands and agrees that as conditions to Mentor Graphics' obligations under this section Customer must: (a) notify Mentor Graphics promptly in writing of the action; (b) provide Mentor Graphics all reasonable information and assistance to settle or defend the action; and (c) grant Mentor Graphics sole authority and control of the defense or settlement of the action.

    11.2. If a claim is made under Subsection 11.1 Mentor Graphics may, at its option and expense: (a) replace or modify the Product so that it becomes noninfringing; (b) procure for Customer the right to continue using the Product; or (c) require the return of the Product and refund to Customer any purchase price or license fee paid, less a reasonable allowance for use.

    11.3. Mentor Graphics has no liability to Customer if the action is based upon: (a) the combination of Software or hardware with any product not furnished by Mentor Graphics; (b) the modification of the Product other than by Mentor Graphics; (c) the use of other than a current unaltered release of Software; (d) the use of the Product as part of an infringing process; (e) a product that Customer makes, uses, or sells; (f) any Beta Code or Product provided at no charge; (g) any software provided by Mentor Graphics' licensors who do not provide such indemnification to Mentor Graphics' customers; or (h) infringement by Customer that is deemed willful. In the case of (h), Customer shall reimburse Mentor Graphics for its reasonable attorney fees and other costs related to the action.

    11.4. THIS SECTION 11 IS SUBJECT TO SECTION 8 ABOVE AND STATES THE ENTIRE LIABILITY OF MENTOR GRAPHICS AND ITS LICENSORS, AND CUSTOMER'S SOLE AND EXCLUSIVE REMEDY, FOR DEFENSE, SETTLEMENT AND DAMAGES, WITH RESPECT TO ANY ALLEGED PATENT OR COPYRIGHT INFRINGEMENT OR TRADE SECRET MISAPPROPRIATION BY ANY PRODUCT PROVIDED UNDER THIS AGREEMENT.

12. **TERMINATION AND EFFECT OF TERMINATION.**

12.1. If a Software license was provided for limited term use, such license will automatically terminate at the end of the authorized term. Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement immediately upon written notice if Customer: (a) exceeds the scope of the license or otherwise fails to comply with the licensing or confidentiality provisions of this Agreement, or (b) becomes insolvent, files a bankruptcy petition, institutes proceedings for liquidation or winding up or enters into an agreement to assign its assets for the benefit of creditors. For any other material breach of any provision of this Agreement, Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement upon 30 days written notice if Customer fails to cure the breach within the 30 day notice period. Termination of this Agreement or any license granted hereunder will not affect Customer's obligation to pay for Products shipped or licenses granted prior to the termination, which amounts shall be payable immediately upon the date of termination.

12.2. Upon termination of this Agreement, the rights and obligations of the parties shall cease except as expressly set forth in this Agreement. Upon termination, Customer shall ensure that all use of the affected Products ceases, and shall return hardware and either return to Mentor Graphics or destroy Software in Customer's possession, including all copies and documentation, and certify in writing to Mentor Graphics within ten business days of the termination date that Customer no longer possesses any of the affected Products or copies of Software in any form.

13. **EXPORT.** The Products provided hereunder are subject to regulation by local laws and United States ("U.S.") government agencies, which prohibit export, re-export or diversion of certain products, information about the products, and direct or indirect products thereof, to certain countries and certain persons. Customer agrees that it will not export or re-export Products in any manner without first obtaining all necessary approval from appropriate local and U.S. government agencies. If Customer wishes to disclose any information to Mentor Graphics that is subject to any U.S. or other applicable export restrictions, including without limitation the U.S. International Traffic in Arms Regulations (ITAR) or special controls under the Export Administration Regulations (EAR), Customer will notify Mentor Graphics personnel, in advance of each instance of disclosure, that such information is subject to such export restrictions.

14. **U.S. GOVERNMENT LICENSE RIGHTS.** Software was developed entirely at private expense. The parties agree that all Software is commercial computer software within the meaning of the applicable acquisition regulations. Accordingly, pursuant to U.S. FAR 48 CFR 12.212 and DFAR 48 CFR 227.7202, use, duplication and disclosure of the Software by or for the U.S. government or a U.S. government subcontractor is subject solely to the terms and conditions set forth in this Agreement, which shall supersede any conflicting terms or conditions in any government order document, except for provisions which are contrary to applicable mandatory federal laws.

15. **THIRD PARTY BENEFICIARY.** Mentor Graphics Corporation, Mentor Graphics (Ireland) Limited, Microsoft Corporation and other licensors may be third party beneficiaries of this Agreement with the right to enforce the obligations set forth herein.

16. **REVIEW OF LICENSE USAGE.** Customer will monitor the access to and use of Software. With prior written notice and during Customer's normal business hours, Mentor Graphics may engage an internationally recognized accounting firm to review Customer's software monitoring system and records deemed relevant by the internationally recognized accounting firm to confirm Customer's compliance with the terms of this Agreement or U.S. or other local export laws. Such review may include FlexNet (or successor product) report log files that Customer shall capture and provide at Mentor Graphics' request. Customer shall make records available in electronic format and shall fully cooperate with data gathering to support the license review. Mentor Graphics shall bear the expense of any such review unless a material non-compliance is revealed. Mentor Graphics shall treat as confidential information all information gained as a result of any request or review and shall only use or disclose such information as required by law or to enforce its rights under this Agreement. The provisions of this Section 16 shall survive the termination of this Agreement.

17. **CONTROLLING LAW, JURISDICTION AND DISPUTE RESOLUTION.** The owners of certain Mentor Graphics intellectual property licensed under this Agreement are located in Ireland and the U.S. To promote consistency around the world, disputes shall be resolved as follows: excluding conflict of laws rules, this Agreement shall be governed by and construed under the laws of the State of Oregon, U.S., if Customer is located in North or South America, and the laws of Ireland if Customer is located outside of North or South America. All disputes arising out of or in relation to this Agreement shall be submitted to the exclusive jurisdiction of the courts of Portland, Oregon when the laws of Oregon apply, or Dublin, Ireland when the laws of Ireland apply. Notwithstanding the foregoing, all disputes in Asia arising out of or in relation to this Agreement shall be resolved by arbitration in Singapore before a single arbitrator to be appointed by the chairman of the Singapore International Arbitration Centre ("SIAC") to be conducted in the English language, in accordance with the Arbitration Rules of the SIAC in effect at the time of the dispute, which rules are deemed to be incorporated by reference in this section. Nothing in this section shall restrict Mentor Graphics' right to bring an action (including for example a motion for injunctive relief) against Customer in the jurisdiction where Customer's place of business is located. The United Nations Convention on Contracts for the International Sale of Goods does not apply to this Agreement.

18. **SEVERABILITY.** If any provision of this Agreement is held by a court of competent jurisdiction to be void, invalid, unenforceable or illegal, such provision shall be severed from this Agreement and the remaining provisions will remain in full force and effect.

19. **MISCELLANEOUS.** This Agreement contains the parties' entire understanding relating to its subject matter and supersedes all prior or contemporaneous agreements. Some Software may contain code distributed under a third party license agreement that may provide additional rights to Customer. Please see the applicable Software documentation for details. This Agreement may only be modified in writing, signed by an authorized representative of each party. Waiver of terms or excuse of breach must be in writing and shall not constitute subsequent consent, waiver or excuse.

Rev. 130502, Part No. 255853

Corporate Headquarters
Mentor Graphics Corporation
8005 SW Boeckman Road
Wilsonville, Oregon 97070-7777
United States of America
503-685-7000

Sales and Product Information
800-547-3000 or 503-685-8000

North American Support Center
800-547-4303

For support locations outside of the US, please refer to the following website: http://www.mentor.com/supportnet/support-offices.html

SSPA star★award hall of fame

www.mentor.com/supportnet